

GRAZIANO

Tortona S.p.A.

FANUC SERIES 21i/18i/16i – TA

Concise guide

Edition 03.01

POSIZIONE ATTUALE				01000 N00000			
(ASSOLUTO)				F 0 MM/M			
X ₁	259.682			MAN F	0	CONT PEZZI	108
Z ₁	50.518			TMP LAV	30H32M	TMP CICLO	0H 0M 4S
C ₁	278.895			PROGRAMMA			
E ₁	576.000			01000 (TEST.POST.D. 20 C6641234) ;			
(MODALE)				M600 ;			
G00	G40	G49	F	M	37	G53 X290 Z300 ;	
G97	G25	G54	S			T1200 ;	
G90	G22	G64	SRPM	0		G0 Y0 ;	
G69	G80	G18	SSPM	0	T	G10 L2 P1 Z177.5 ;	
G95	G98	G13.1	SMAX	32767		M1 ;	
G21	G67		SACT	0		M601 ;	
				M602 ;			
				>^			
				OS 50% T0000			
				EDIT ***** 10:29:35 CN1			
ASSOL+ REL TUTTO				PROGR DIR (OPER) +			

0.1 GENERAL INDEX- CONCISE GUIDE FOR PROGRAMMER

PAGE	PAR.	CONTENTS
7	1.0	FOREWORD
8	2.0	NC MAIN FUNCTIONS AND ADDRESSES
8	2.1 O	Program and sub-program number
8	2.2 N	Block number
9	2.3 G	Preparatory operations
9	2.4 X/Z/B/Y	Movement absolute co-ordinates
10	2.5 U/W	Movement incremental co-ordinates
12	2.6 F	Work feed
12	2.7 S	Spindle rotation speed
13	2.8 T	Tool selection
15	2.9 M	Auxiliary functions
18	2.10 M	Other auxiliary functions
19	2.11 /	Skipping a block
19	2.12 ()	Notes and comments
20	3.0	ISO PROGRAMMING
20	3.1 G0	Linear axes rapid traverses
21	3.2 G1	Work linear interpolation
24	3.3 G1 A..	Programming with angles
28	3.4 G2/G3	Circular interpolations
30	3.5 G4	Axis pause time
31	3.6 G95	Feed in mm/rev
31	3.7 G94	Feed in mm/min
32	3.8 G97	Fixed revolutions spindle rotation
33	3.9 G96	Constant cutting speed
34	3.10 G92	Spindle revolution limitations
35	3.11 G33	Thread cutting movements
37	3.12 G41/G42/G40	Tool radius compensation
41	3.13 G54/G59	Workpiece origins
43	3.14 G52	Origin transfer by program

PAGE	PAR.	CONTENTS
44	3.15 M134/M135	Precise stop
45	3.16 G	List of main “G” preparatory functions
47	4.0	FIXED FANUC CYCLES
47	4.1 G71	Material removal by turning
53	4.2 G72	Material removal by facing
57	4.3 G73	Profile repetition
60	4.4 G70	Finishing cycle
63	4.5 G174	Radial grooves rough machining/pre-finishing cycle
67	4.6 G176	Axial grooves rough machining/pre-finishing cycle
72	4.7 G175/G177	Finishing cycle for radial/axial grooves
76	4.8 G76	Thread cutting cycle in several cuts
81	4.9 G83	Front drilling cycle
83	4.10 G84	Front tapping cycle
85	5.0	SUB-PROGRAMS / PARAMETRIC PROGRAMMING
85	5.1 M98 M99	Use of sub-programs
89	5.2 #	Parametric programming
93	6.0	BACK SPINDLE MACHINING
93	6.1	Most important addresses used
94	6.2 M	Auxiliary functions
95	6.3	Example of machining with back spindle
98	6.4 O9100	Workpiece change-over with parting off
101	6.5 O9101	Workpiece change-over with parting off without extraction
104	6.6 O9102	Workpiece change-over without parting off
106	7.0	AXIS C MACHINING AND MOTOR DRIVEN TOOLS
106	7.1	Motor driven tools
108	7.2	Motor driven tools reset
109	7.3 M37	Axis “C”
110	7.4	Programming in real co-ordinates
111	7.5 M20/M21	Use of spindle brake
112	7.6 G83	Front drilling cycle
115	7.7 G87	Radial drilling cycle

PAGE	PAR.	CONTENTS
118	7.8 O9103	Front tapping sub-program
121	7.9 O9104	Radial tapping sub-program
124	7.10 G112	Programming in imaginary co-ordinates
127	7.11 G2/G3	Circular interpolation in G112
129	7.12 G41 G42 G40	Milling radius offset in G112
131	7.13 G107	Cylindrical interpolation
135	7.14	Programming with real Y axis
138	8.0	BAR MACHINING
138	8.1	Example of machine tool loader use with back spindle
140	8.2	Example of machine tool loader use without back spindle
142	8.3	Ex. of machine tool push-bar conveyor use with back spindle
143	8.4	Ex. of machine tool push-bar conveyor use without back spindle
144	8.5	Example of pull-bar conveyor use

OPERATOR READY REFERENCE GENERAL INDEX


146	12.0	MACHINE START UP
146	12.1	Power-on
146	12.2	Execution of axes reference
146	12.3	Write protection key
147	13.0	PROGRAMME MANAGEMENT
147	13.1	How to create a new programme
147	13.2	How to modify an existing programme
147	13.3	How to insert a code (or a block) in a programme
147	13.4	How to modify or replace a code
148	13.5	How to delete a code
148	13.6	How to delete a block
148	13.7	How to copy /paste part of a programme
148	13.8	How to copy a programme
149	13.9	How to delete a programme
149	13.10	How to rename a programme
149	13.11	Selection of a programme for machining

PAGE	PAR.	CONTENTS
150	13.12	Creation of a new subprogram
150	13.13	Graphic simulation of a programme
151	13.14	Running of the programme in automatic cycle
151	13.15	Interruption of programme execution
151	13.16	How to start the programme from an intermediate stage
151	13.17	Background editing
152	14.0	TOOL RESET
152	14.1	Manual tool reset
153	14.2	Centre reset
153	14.3	Internal machining tools reset
153	14.4	Tool reset on subspindle
153	14.5	Tool reset with probe (optional)
154	14.6	Tool reset for TWIN machines
155	14.7	Tool table management
155	14.8	Tool fine correction
155	14.9	Entry of insert radius
156	14.10	Entry of tool orientation
156	14.11	Entry of cutter radius
157	15.0	ORIGIN MANAGEMENT
157	15.1	Origin measurement
157	15.2	Origin modification
158	16.0	MACHINE PARAMETERS
158	16.1	How to modify a machine parameter
159	17.0	SETTING OF CTX300 TAILSTOCK
159	17.1	Instructions to be inserted in the program
159	17.2	Tailstock double speed option
160	17.3	Tailstock repositioning
161	18.0	CTX SERIES TAILSTOCK AND REST
161	18.1	Manual movement of tailstock and rest
161	18.2	Instructions to insert in program

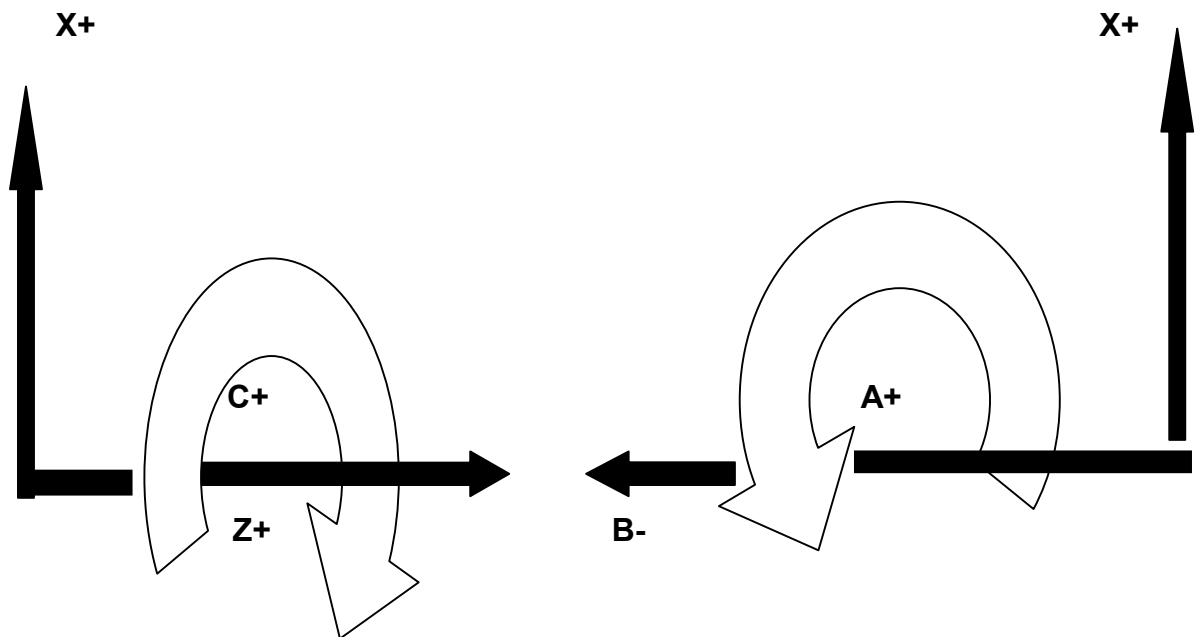
PAGE	PAR.	CONTENTS
163	19.0	KEYBOARD AND OPERATOR'S PANEL
163	19.1	Description of keys on the operator's panel
167	19.2	Description of keys on the MDI panel
170	19.3	Selector switch and keys below the operator's panel
172	20.0	SERIAL PORT COMMUNICATION
172	20.1	Setting of data transfer parameters
172	20.2	Cable scheme
174	20.3	Transmission programs
176	20.4	How to copy a programme to the serial port
176	20.5	How to copy a programme from the serial port
176	20.6	How to copy a programme to memory card
177	20.7	How to copy a programme from memory card

1.0 FOREWORD

On an NC machine tool the sequence of the instructions programmed to process a workpiece consists of codes which are made up of functions or addresses with a relevant numeric value.

When preparing a part program the tool path is imagined referring to a system of co-ordinates, the origin of which ( => zero point to which all the dimensions refer) can be chosen. In the specific case of an NC lathe this co-ordinates system is composed of two or more axes:

- axis X (for diameters).
- axis Z (for lengths).
- axis C (for angle divisions on lathes with controlled spindle).
- axis B (for the longitudinal position of the back spindle on machines fitted with this option).
- axis A (for angle division on lathes with controlled back spindle).



The tool path is programmed with co-ordinated points written in the correct sequence and established according to the workpiece profile. Each movement of the tool along this path is written as a separate instruction (block) together with any technological data required. The group of blocks forms the "PART PROGRAM".

2.0 NC MAIN FUNCTIONS AND ADDRESSES

The sequence of instructions that make up the program consists of letters and numbers, each of which has a specific significance.

2.1 “O” PROGRAM AND SUB-PROGRAM NUMBER

The letter O followed by a number indicate the programs and the sub-programs. The number paired with the letter O can range from 1 to 9999. To have better program management Graziano suggests that the following values are paired :

From **O1** to **O8000 Main Programs** available for the customer

From **O8001** to **O8999 Sub-programs** available for the customer

From **O9000** to **O9999 Sub-programs** available for GRAZIANO to create special macros that cannot be modified by the customer since they are protected by a parameter.

The NC memory can contain a maximum of 63, between Programs and Sub-programs, or a maximum of 32000 characters.

2.2 “N” BLOCK NUMBER

A block is a group of words that identify the operation to be carried out.

Example:

N10 G0 X200 Z5 M8

Each block is identified by a sequential number N, from 0 to 9999 and must end with the end of block EOB character (;)

The block number is entered automatically by the NC when an end of block EOB code is inserted (;).

Through a machine datum (N. 3216) the increment value can be selected in the block numbering: unitary (N1 N2 N3 etc.) or decimal (N10 N20 N30 etc.)

It is up to the programmer whether to use the block number or not.

To use the block number a value 1 has to be assigned to the setting datum SEQUENCE NO. in the Prepare/Manual menu which can be entered by pressing the SETTING key on the MDI keyboard

Usually the block numbering is not enabled.

2.3 “G” PREPARATORY OPERATIONS

The **G** code prepares the control to carry out certain operations that differ according to the number that follows this code (e.g.: G0, G1, G3, etc.).

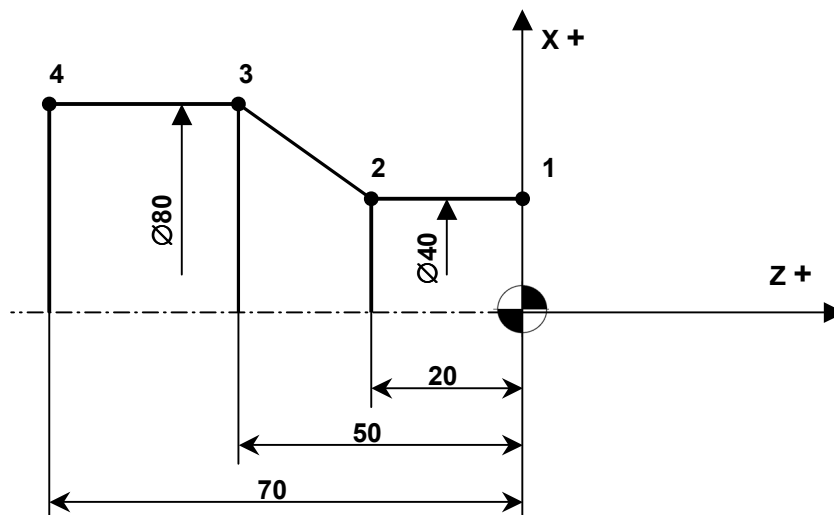
There are two types of preparatory functions: **modal** functions and **self-deletion** functions. The former remain active until they are cancelled by other modal functions, the latter are only active in the block where they are entered.


2.4 “X Z B Y” MOVEMENT ABSOLUTE CO-ORDINATES

Codes **X** and **Z** define the absolute co-ordinates referring to the workpiece zero. **X** determines diameters (diametrical programming); **Z** determines the lengths; **B** determines the back spindle axis movements (only on machines where this option is installed); **Y** determines the motor driven turret Y axis movements (only on machines where this option is installed) .

These codes can be programmed with a positive or a negative sign. If no sign is programmed the value is considered positive. Values can be programmed with up to three digits after the decimal point.

Example:

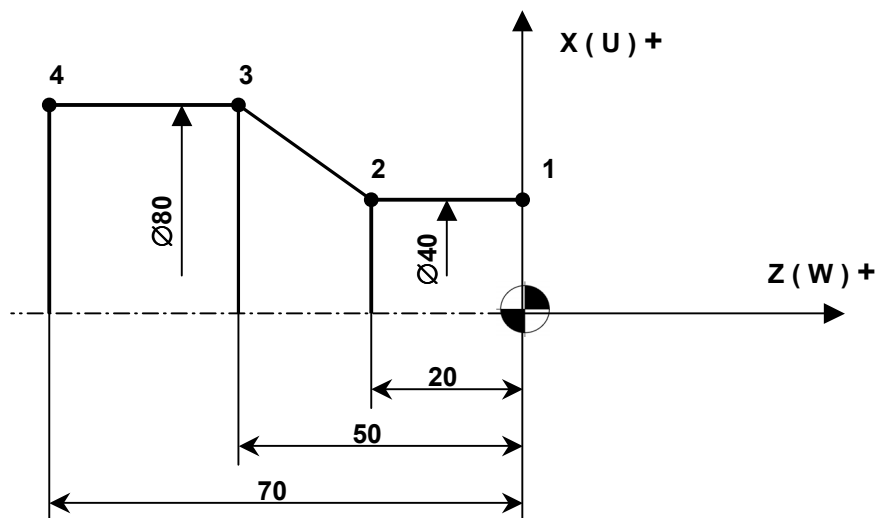



<u>X / Z Co-ordinates</u>	<u>Position</u>
N5 X0 Z0	
N6 X40	(1)
N7 Z-20	(2)
N8 X80 Z-50	(3)
N9 Z-70	(4)

2.5 “U and W” MOVEMENT INCREMENTAL CO-ORDINATES

Codes **U** and **W** define the incremental co-ordinates referring to the last programmed point. **U** defines a movement on axis X (diametrical programming); **W** defines a movement on axis Z. These codes can be programmed with a positive or a negative sign. If no sign is programmed the value is considered positive. Values can be programmed with up to three digits after the decimal point.

Example:



<u>U / W Co-ordinates</u>	<u>Position</u>
N5 X0 Z0	
N6 U40	(1)
N7 W-20	(2)
N8 U40 W-30	(3)
N9 W-20	(4)

The first program start value and the first position of each tool must always be programmed in absolute co-ordinates. It is possible to program an absolute co-ordinate and an incremental co-ordinate in the same block, providing they do not refer to the same axis.

Example:

N10 G0 X100 W-5 ; correct

N10 G0 U10 Z100 ; correct

N30 G0 X100 U20 ; not correct

2.6 "F" WORK FEED

Function **F** (Feed) defines the work feed and can have two different significances, according to which preparatory G function is active (G95 or G94 see par. 3.6 and par. 3.7):

- **mm/rev** (usually used for turning operations).
- **mm/min** (usually used for milling operations or for work movements with spindle stationary).

The programmed feed **F** can be modified through the axis trimmer with a variable value from 0% to 120%.
The programmed feed **F** remains active until another is selected.

2.7 "S" SPINDLE ROTATION SPEED

Function **S** (Speed) defines the rotation speed of the spindle. It can have two different significances, according to which preparatory G function is active (G97 or G96 see par. 3.8 and par. 3.9):

- **rpm** (usually used for machining without wide diameter variations e.g.: drilling, tapping and thread cutting).
- **m/min** (usually used for all turning operations).

The programmed speed can be changed through the spindle trimmer with a variable value from 50% to 120%.

2.8 “T” TOOL SELECTION

Code **T** (Tool) defines the tool corrector and the position of the turret to be activated for machining. The tool corrector contains information that identifies the characteristics (length, direction, radius etc.) of the tool. When programming, the tool setting is always composed of 3 or 4 digits. The first number, or first pair of numbers, defines the position of the tool in the turret; this number is therefore usually between 1 and 12.

The second pair of numbers, always composed of two digits, identifies the corrector matched to the tool. The control memory usually has available 32 tool correctors; therefore the programmer has to select the corrector to match to each individual tool.

For simpler operation it is suggested to match a tool number to the same corrector number.

Example:

N1 **T101**

N2

N3

N4

N5

N6

N7 **T404**

N8

N9

N10

N11



Machining with tool T1 corrector 01
1



Machining with tool T4 corrector 04

Under certain circumstances it is possible to match a tool with a different corrector, for example to move the position of a tool in the turret without having to reset it again.

Example:

N4 **T121**

; Tool selection T1 with corrector 21

N5

N6

N7

N8



Machining

When a tool is called up, the turret rotates so as to follow the shortest path, whether clockwise or anti-clockwise.

In the machines provided with hydraulic turrets, there are two functions to select the desired turret rotation direction. These functions are **M16** and **M46**.

M16 forces the clockwise rotation of the turret disk.

M46 forces the anti-clockwise rotation of the turret disk.

Example:

N3

N4 **T101** ; Tool selection T1 shortest path

N5

N6 **T303 M16** ; Tool selection T3 in clockwise rotation

N7

N8 **T606 M46** ; Tool selection T6 in anti-clockwise rotation

N9

In some cases it may be useful to make movements without any corrector active or rather, without taking into account the tool length, for example to bring the turret in the smallest overall dimension zone when using automatic loaders or such like. The function that disables the tool correctors is **T0**. To reactivate the correctors it is sufficient to call up a tool.

T0 does not rotate the turret disk.

2.9 “M” AUXILIARY FUNCTIONS

Auxiliary functions are used to send commands to the control and to the machine tool and they are divided between functions that become operational as soon as they are read, and functions that become operative at the end of block (M0, M1, M3, M4)..

The list below indicates the most commonly used M auxiliary functions :

M0 => Stop program . Interrupts the program running and stands by until it receives consent to continue from the operator (start cycle).

M1 => Stop program-optional. When active it interrupts the program running and stands by until it receives consent to continue from the operator (start cycle).

To activate this command see paragraph 19.1

M3 => Clockwise spindle rotation. The spindle rotates clockwise at the previously set speed S.

M4 => Spindle anti-clockwise rotation. The spindle rotates anti-clockwise at the previously set speed S

M5 => Spindle rotation stop. This function stops the spindle rotation

M8 => Open coolant. This function activates the delivery of the coolant. The spindle rotation influences the function activation: if the spindle is not rotating the coolant delivery is deactivated.

M9 => Stop coolant. This function stops the delivery of the coolant.

M13 => Spindle clockwise rotation at previously set speed S and coolant delivery activated.

M14 => Spindle anti-clockwise rotation at previously set speed S and coolant delivery activated.

M19 => Spindle orientation. This function stops the spindle in a defined angle position. M19 can be programmed also with the spindle rotating. The stopping angle is programmed through the optional address **S**. The M5 function must always be programmed after this function .

Example: N22

 N23 **M19 S45**

 N24 **M5**

 N25

M30 => End of program. This function terminates the running of the program and sets the NC to start from the first block.

The **M** functions listed below are used for many specific applications. Details regarding the use of these functions can be found in the machine documentation.

- M0** ⇔ stop program
- M1** ⇔ optional stop program
- M2** ⇔ end of program (without re-winding)
- M3** ⇔ spindle clockwise rotation
- M4** ⇔ spindle anti-clockwise rotation
- M5** ⇔ stop spindle
- M7** ⇔ coolant delivery not depending on spindle rotation
- M8** ⇔ coolant delivery depending on spindle rotation
- M9** ⇔ cut off coolant
- M10** ⇔ air blast activation to clean jaws (spindle rotation enabled with jaws open)
- M11** ⇔ deactivation of jaw cleaning air blast (spindle rotation disabled with jaws open)
- M12** ⇔ reduction of self-centring chuck locking pressure
- M13** ⇔ spindle clockwise rotation and coolant delivery
- M14** ⇔ spindle anti-clockwise rotation and coolant delivery
- M16** ⇔ force turret clockwise direction (only for hydraulic turrets)
- M18** ⇔ restore normal pressure to self-centring chuck lock
- M19** ⇔ spindle direction (M19 Sxx directs the spindle to xx degrees)
- M20** ⇔ spindle brake on
- M21** ⇔ spindle brake release
- M22** ⇔ tailstock sleeve forward feed with conditioning
- M23** ⇔ tailstock sleeve backward movement with conditioning
- M24** ⇔ tailstock sleeve forward feed without conditioning
- M25** ⇔ tailstock sleeve backward movement without conditioning
- M26** ⇔ automatic sliding guard opening
- M27** ⇔ automatic sliding guard closing
- M30** ⇔ end of program (with winding)
- M31** ⇔ conditionings suspended on next tool change
- M32** ⇔ steady rest release from bench and hooking onto carriage
- M33** ⇔ steady rest release from carriage and hooking onto bench
- M36** ⇔ axis C disengagement
- M37** ⇔ axis C engagement
- M38** ⇔ tool reset sensor in working position
- M39** ⇔ tool reset sensor in home position
- M46** ⇔ force turret anti-clockwise direction (only for hydraulic turrets)

-
- M52** ⇨ tailstock release from bench and hooking onto carriage
 - M53** ⇨ tailstock release from carriage and hooking onto bench
 - M58** ⇨ spindle and reset sensor orientation in work position
 - M62** ⇨ workpiece counter increment on display (only active in automatic mode)
 - M63** ⇨ external robot call to change workpiece (optional)
 - M64** ⇨ workpiece released indication to external robot (optional)
 - M65** ⇨ workpiece locked indication to external robot (optional)
 - M67** ⇨ command / wait for bar change to loader (optional)
 - M68** ⇨ self-centring chuck /collet chuck closure
 - M69** ⇨ self-centring chuck /collet chuck opening
 - M74** ⇨ second steady rest arms opening (optional)
 - M75** ⇨ second steady rest arms closing (optional)
 - M78** ⇨ bar measurement check (option) for Irco loader
 - M79** ⇨ bar at end of stroke check (option) for Irco loader
 - M84** ⇨ steady rest arms opening
 - M85** ⇨ steady rest arms closing
 - M86** ⇨ retractable steady rest in working position (up)
 - M87** ⇨ retractable steady rest in home position (down)
 - M88** ⇨ workpiece unloading arm in home position (down)
 - M89** ⇨ workpiece unloading arm in working position (up)
 - M90** ⇨ probe parameters memorisation at PMC (from #812 to #822)
 - M100** ⇨ temporary suspension of active S

2.10 “M” OTHER AUXILIARY FUNCTIONS

The list below indicates other **M** functions used for many specific applications. Details regarding the use of these functions can be found in the machine documentation.

- M29** ⇨ rigid tapping on spindles (cannot be used with motor driven tools)
- M98** ⇨ sub-program call up (M98 P...)
- M99** ⇨ return from sub-program
- M127** ⇨ deactivates M128/M129/M130 and immediately stops the conveyor
- M128** ⇨ conveyor pulsed movement in cycle (counter C11 in minutes)
- M129** ⇨ conveyor intermittent movement in cycle (counter C10/C11 in minutes)
- M130** ⇨ conveyor continuous movement in cycle
- M131** ⇨ turret pre-release
- M922** ⇨ sleeve thrust enabled
- M923** ⇨ sleeve thrust suspended (if thrust switch is set to 1)
- M950** ⇨ self-centring chuck pedal disabled
- M951** ⇨ self-centring chuck pedal re-enabled
- M966** ⇨ spintor feed suspend (for barfeeder IEMCA)
- M967** ⇨ spintor feed restart (for barfeeder IEMCA)
- M968** **barfeeder thrust suspend**
- M969** ⇨ push-bar conveyor thrust restored
- M970** ⇨ push-bar conveyor use disabled
- M971** ⇨ push-bar conveyor use restored
- M984** ⇨ external workpiece pick-up (shafts)
- M985** ⇨ internal workpiece pick-up (flanges)
- M995** ⇨ emergency light on
- M999** ⇨ machine tool cut off by program (NC remains on)

2.11 “ / “ SKIPPING A BLOCK

This function is used to run or exclude the marked block.

To activate or exclude this function use the relevant key on the operator panel (“see paragraph 19.1)

- With the key warning light **off** the barred blocks are **run**.
- With the key warning light **on** the barred blocks are **skipped**.

Example:

```
N10 /T101
N20 /G54
N30 /G92 S2000
N40 /G96 S180 M4
N50 /G0 X100 Z2 M8
N60 /G1 Z-40 F0.25
```

2.12 NOTES AND COMMENTS

For programming requirements comments and notes can be entered into the program, for example an indication of the type of tool next to the block where that tool is selected.

These notes can be entered in round brackets (...)

- (...) a note written in round brackets can contain up to 30 characters, and is visible both during programming and when the program is run

Example:

```
N10 T101 (EXTERNAL ROUGH MACHINING TOOL)
```

or

```
N18 M0 (TURN THE WORKPIECE)
```

3.0 ISO PROGRAMMING

ISO language is a unified programming system common to many controls on different types of machine tools of different nature.

3.1 “G0” LINEAR AXES RAPID TRAVERSES

The “G0” function controls rapid axis movement (at maximum speed). This function is used to separate from or approach the workpiece at a safe distance. This block must contain one or more destination coordinates (X e Z).

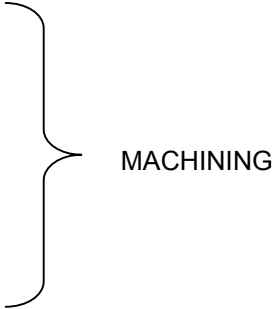
Programming “G0 X... Z...” the tool starts from its current position and reaches that programmed in a linear movement (thus following the route).

“G0” remains modularly active until another movement of the same group (G1, G2, G3) is performed.

The G0 function is therefore used to approach the workpiece at the beginning of machining and to separate from it at the end of cycle.

Example:

```
N17 .....  
N18 G0 X50 Z2 ; rapid traverse  
N19 .....  
N20 .....  
N21 .....  
N22 .....  
N23 .....  
N24 .....  
N25 .....  
N26 .....  
N27 .....  
N28 G0 X200 Z100 ; rapid return  
N29 .....
```



3.2 “G1” WORK LINEAR INTERPOLATION

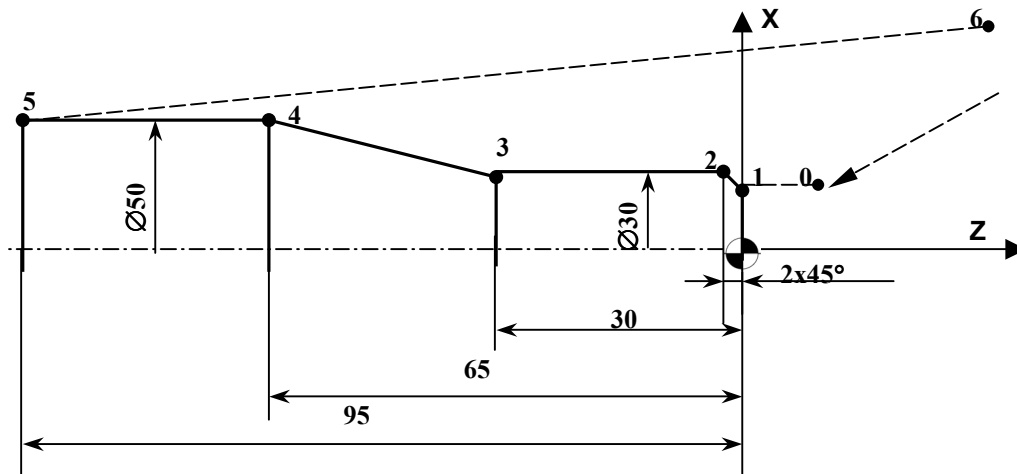
The “G1” function controls a linear work movement (at a programmed speed). This function is used to carry out machining on the workpiece.

With this function it is the programmer who decides the speed (feed “F”) at which the tool is to reach the programmed point. The same block must also contain one or two destination co-ordinates (X and Z) and the feed (F) if this has not been inserted beforehand.

Programming “G1 X... Z... F...” the tool starts from its current position and reaches that programmed in a linear movement at the work speed.

Function “G1” and work feed “F” are modal functions.

Example:



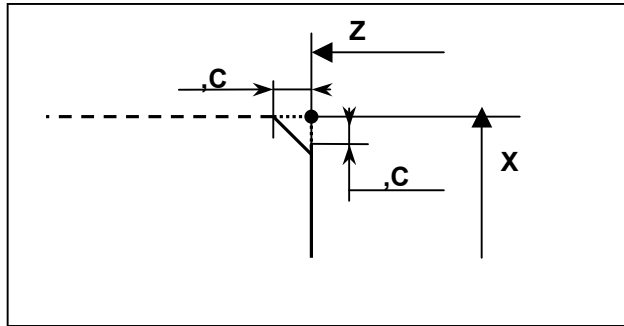
N1		
N2 G0 X26 Z3	(0)	Approach
N3 G1 Z0 F0.2	(1)	Turning
N4 X30 Z-2	(2)	
N5 Z-30	(3)	
N6 X50 Z-65 F0.1	(4)	
N7 Z-95	(5)	
N8 G0 X100 Z30	(6)	Separation
N9		

The linear movement programmed with G1 can be linked to the movement of the next block by a chamfer (,C) or a connecting radius (R).

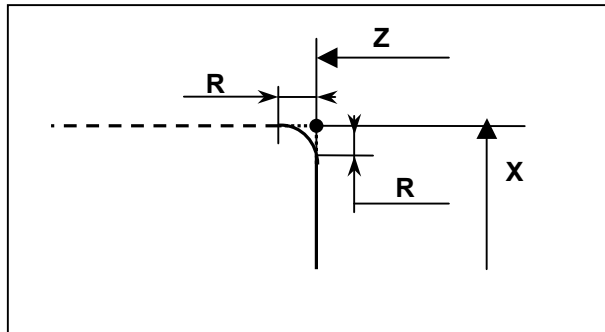
For two-axis machines (without the axis C option) the chamfer can be identified by just the letter C followed by the value (and not by ,C)

Example:

N12
N13 G1 X... Z... ,C...
N14

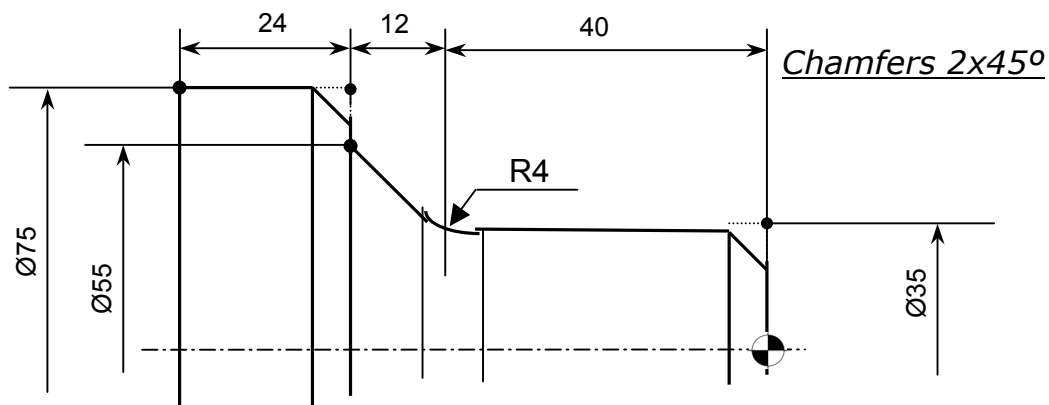


N12
N13 G1 X... Z... R...
N14



These functions can only be programmed in a "G1" block. It is also important to underline that the block following one containing "R" or ",C" must be a G1 work movement so that the chamfer or radius can be calculated by the control.

Example of how to use the R and ,C functions:



N5

N6 G0 X0 Z3

Approach

N7 G1 Z0 F0.2

N8 X35 ,C2

N9 Z-40 R4

N10 X55 Z-52 F0.1

N11 X75 ,C2

N12 Z-76

N13 G0 X100 Z50

Separation

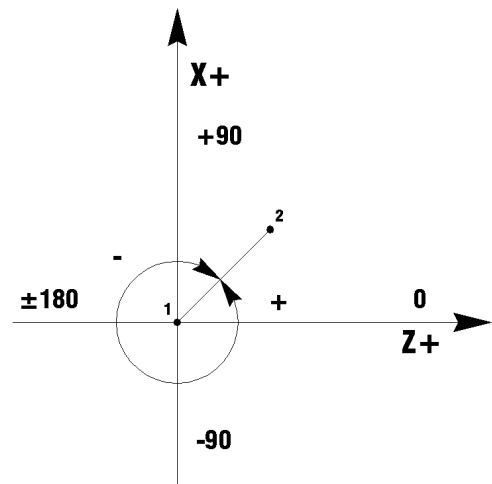
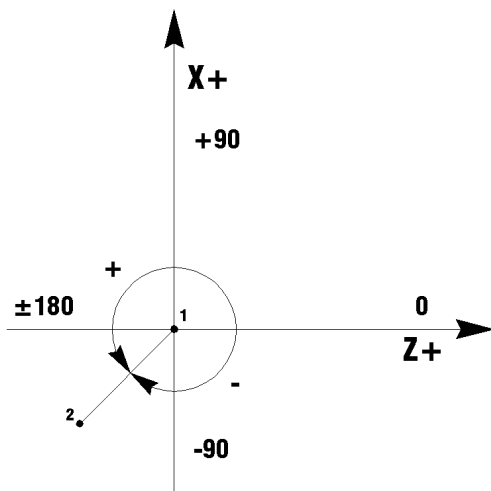
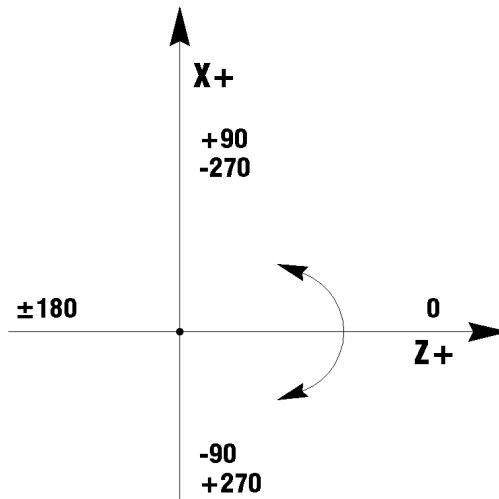
N14

Profile description

3.3 “G1 A...” PROGRAMMING WITH ANGLES

When using G1 instructions as well as the end of movement co-ordinates X and/or Z, besides radii or chamfers on final points (R and ,C) the programmer can indicate the movement angle (A or ,A on machines that have the motor driven back spindle option)

When programming the angle, value A can be positive or negative in a range from 0° to 360°. To define the angle value, see the schematised figure imagining to position the “cross” with the centre on the first point of the straight line. The angle of the line is determined by imagining to turn the cross zero (axis Z) in the positive or negative direction to meet the straight line.



The use of the A angle makes it possible to program just one final point matched to the movement angle instead of two final points (X e Z), or in certain conditions, to insert only the line angle without any final co-ordinate.

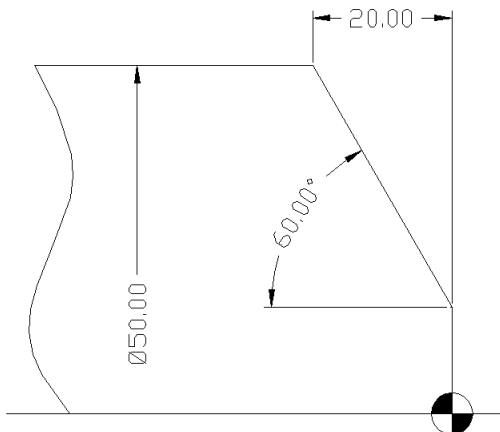
Therefore there are two possibilities :

G1 X... A... (final point in X and angle) with any chamfers (,C) or radii (R) on the final point

G1 A... (angle only) with any chamfers (,C) or radii (R) on the final point

If only G1 A is used, the next block must absolutely contain both final co-ordinates (X, Z) and the angle (A) with eventual chamfers (, C) or radius (R) on final point.

Example :



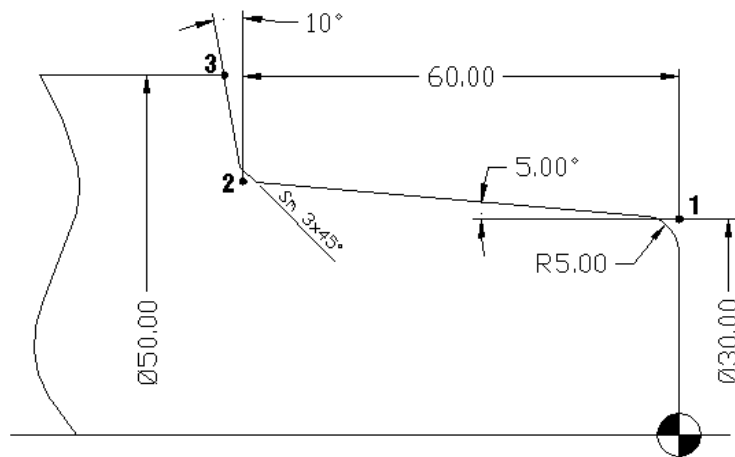
```
N48 G0 X0 Z2
N49 G1 Z0 F0.25
N50 G1 ,A90
N51 G1 X50 Z-20 A120
```

The value of angle A must be in centesimal degrees brought to the third decimal digit

Example :

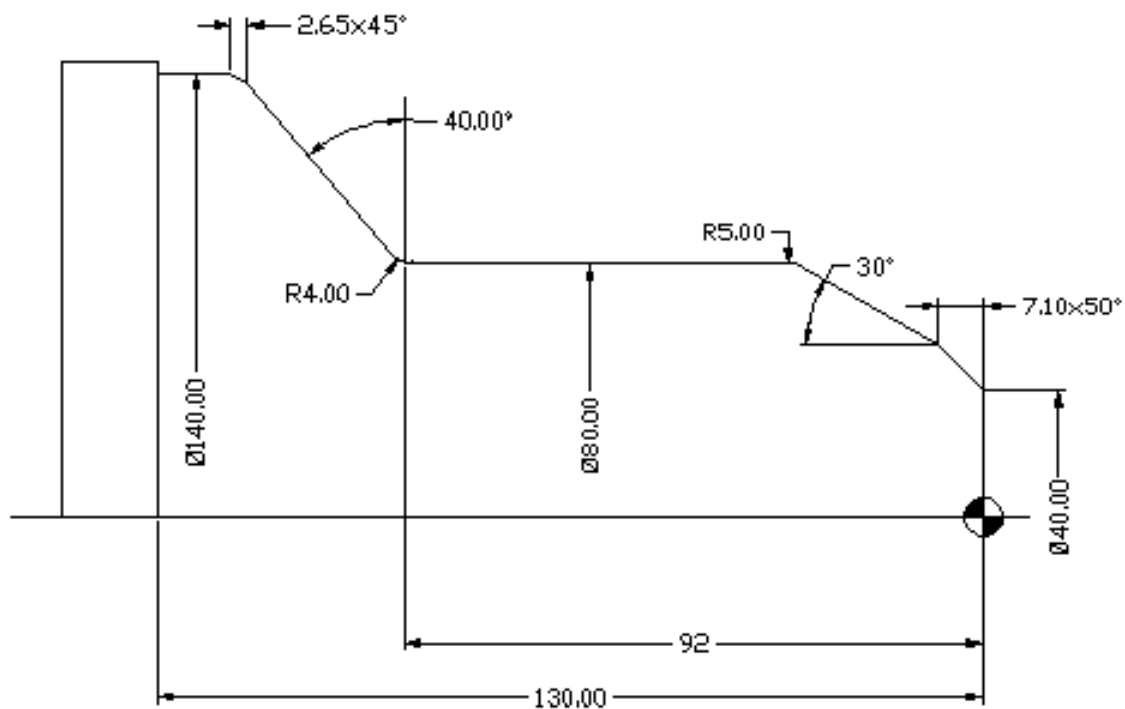
```
N55 G1 ,A15.123
```

Example of programming using the angles:



```
N48 G0 X0 Z2  
N49 G1 Z0 F0.25  
N50 X30 R5  
N51 Z-60 ,A175 ,C3  
N52 X50 ,A100  
N53 G0 X200 Z200
```

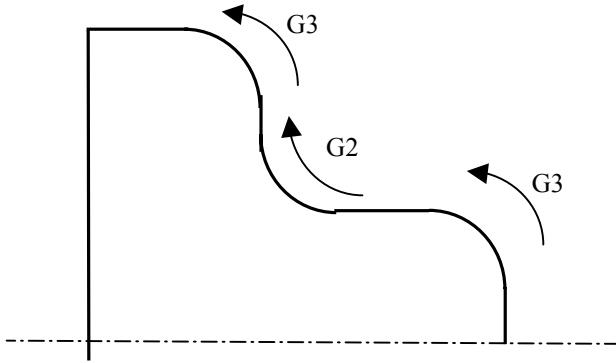
Example of programming using the angles:



N48 G0 X0 Z2
N49 G1 Z0 F0.25
N50 X40
N51 Z-7.1 ,A130
N52 X80 ,A150 R5
N53 Z-92 R4
N54 X140 ,A130 ,C2.65
N55 Z-130
N56 X160
N57 G0 X200 Z200

3.4 “G2 / G3” CIRCULAR INTERPOLATIONS

Functions G2 and G3 are programmed to make circle arcs in clockwise or anti-clockwise direction as shown in the figure:



The block with circular interpolation is programmed:

```
N24 G2 X... Z... R...           ; Clockwise
N31 G3 X... Z... R...           ; Anti-clockwise
```

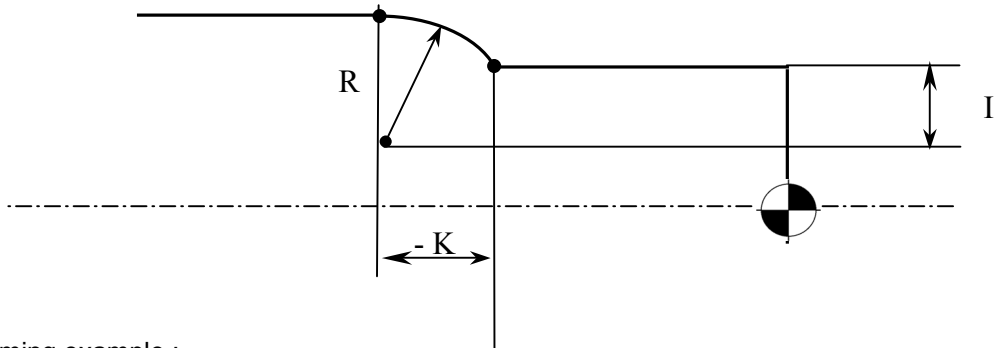
Or:

```
N15 G2 X... Z... I... K...       ; Clockwise
N18 G3 X... Z... I... K...       ; Anti-clockwise
```

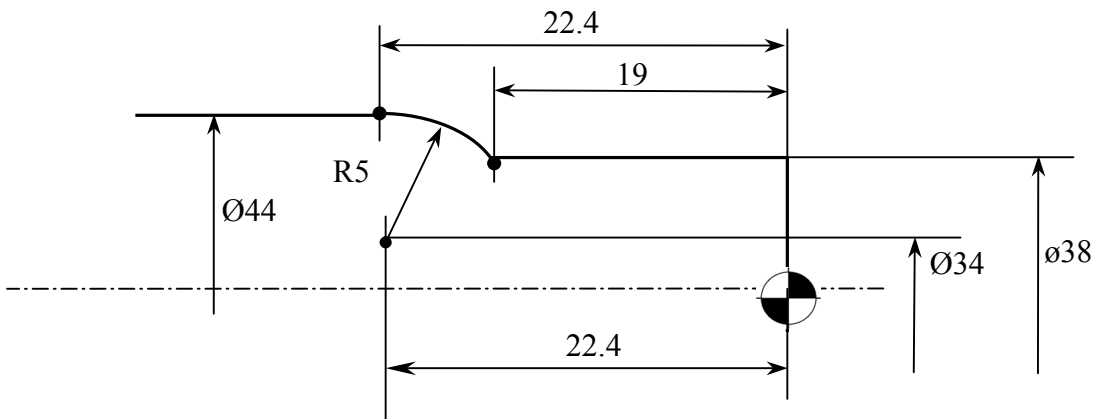
Where:

- G2 / G3 => Direction of circular interpolation
- X => Co-ordinate of final point along axis X
- Z => Co-ordinate of final point along axis Z
- R => Radius of circular interpolation
- I => Incremental distance of starting point at the radius centre of the interpolation along axis X (radial value)
- K => Incremental distance of starting point radius at the centre of the interpolation along axis Z

I and K functions trend :



Programming example :



N5
 N6 G0 X38 Z3
 N7 G1 Z-19 F0.2
 N8 G3 X44 Z-22.4 R5
 N9 G1 Z-30
 N10

Or:

N5
 N6 G0 X38 Z3
 N7 G1 Z-19 F0.2
 N8 G3 X44 Z-22.4 1-2 K-3.4
 N9 G1 Z-30
 N10

G2 and G3 are modal functions and are cancelled by programming a linear movement G function (G0, G1).

3.5 “G4” AXIS PAUSE TIME

The G4 function controls a machine axes pause during the running of a cycle for a time, indicated in seconds, that can be programmed with address U.

The G4 can be thus programmed:

N12

N13 **G4 U1**

N14

Where :

- **G4** => Activates the pause of the machine axes.
- **U** => Defines the time of the axes pause in seconds.
 Minimum value 0.001 seconds, maximum value 9999.999 seconds.

Function G4 is self deleting therefore it automatically disables in the block following the one where it is located.

Always indicating the pause in seconds, it is also possible to have the pause in number of revolutions by using this formula :

Seconds of pause for one spindle revolution = $60 / S$ (spindle speed in rpm)

Example:

If the spindle rotates at 300 rpm, the pause time for one revolution will be $60 / 300 = 0.2$ seconds

If a pause is required equal to 3 rpm, write : **G4 U0.6** (0.2 seconds x 3 rpm)

3.6 “G95” FEED IN MM/REV

The **G95** function selects the feed **F** in **mm/rev**. When this function is active the feed values will be programmed as follows: F0.05, F0.15, F0.3, F0.5 and so forth. **G95** is automatically activated when the machine is switched on, therefore it is not necessary to specify its activation in the program. It is a **modal** function and can be cancelled by programming code G94.

```
N4 .....  
N5 G1 Z-30 F0.3      ; Program with G95 (F= mm/rev.) present at power on  
N6 .....  
N7 .....  
N8 .....  
N9 G94              ; Program with G94 (F= mm/min)  
N10 G1 Z50 F500  
N11 .....  
N12 G95             ; Program with G95 (F= mm/rev.)  
N13 G1 Z-20 F0.2  
N14 .....
```

3.7 “G94” FEED IN MM/MIN

The **G94** function selects feed **F** in **mm/min**. When this function is active the feed values will be programmed as follows: F50, F150, F500, F2000 and so forth. This function is used to perform movements with work feed when the spindle is stationary, or when it is necessary to release the axis feed from the spindle revolutions (e.g.: when milling with motor driven tools). G94 is a **modal** function and can be cancelled by programming the code G95.

```
N5 G1 X... Z... F0.2      ; Feed mm/rev. (present at power on)  
N6 .....  
N7 .....  
N8 G94              ; mm/min feed set  
N9 G1 X... Z... F400  
N10 .....  
N11 .....  
N12 G95             ; mm/rev feed set  
N13 G1 X... Z... F0.12  
N14 .....
```

3.8 “G97” FIXED REVOLUTIONS SPINDLE ROTATION

Function **G97** prepares the spindle speed in revs/min (fixed revs) set by the code **S**. When this function is active the programmed S value represents the actual number of revolutions per minute of the spindle. (e.g.: S50, S160, S500, S1200, S3200, S5000 etc.). G97 is automatically activated when the control is switched on, therefore it is not necessary to specify its activation in the program. It is a **modal** function and can be cancelled by programming G96 (cutting speed set Vt [m/min.]).

This function is recommended when drilling and thread cutting, and is necessary for tapping. Programming an S value with G97 active, and knowing the working diameter, the cutting speed value can be calculated using this formula:

$$V_t = \frac{\pi \times D \times n}{1000} \quad \text{Where} \quad \left\{ \begin{array}{ll} V_t & \Rightarrow \text{cutting speed [m/min]} \\ \pi & \Rightarrow 3.14 \\ D & \Rightarrow \text{work diameter} \\ n & \Rightarrow \text{rpm} \\ 1000 & \Rightarrow \text{m to mm conversion} \end{array} \right.$$

To calculate the cutting speed for machining performed at 1500 rpm on a diameter of 40:

$$\left. \begin{array}{ll} V_t & = ? \text{ [m/min.]} \\ \pi & = 3.14 \\ D & = 40 \text{ mm} \\ n & = 1500 \text{ rpm.} \end{array} \right\} \quad V_t = \frac{3.14 \times 40 \times 1500}{1000} = 188.4$$

A block containing G97 is programmed:

N4 T101

N5 **G97 S1500 M4**

N6 G0 X100 Z3 M8

Where:

- **G97** => Spindle speed set in rpm
- **S1500** => Number of spindle rpm
- **M4** => Spindle direction of rotation

3.9 “G96” CONSTANT CUTTING SPEED

G96 sets the spindle rotation indicated by the code **S** as constant cutting speed (m/min). With this function active the programmed S value is the surface speed in metres per minute (e.g.: S80, S100, S120, S200, S350 etc.), this function continuously updates the actual spindle revolutions according to the work diameter, keeping the cutting speed constant. It is a **modal** function and can be cancelled by programming G97 (rpm set).

During the turning operations (rough machining, finishing,) it is recommended to always use G96; the S values to be set depend on the type of material, the type of tool, the machining method and so forth.

Example:

N4 T303

N5 **G96 S180 M4**

N6 G0 X100 Z3 M8

Programming an S value with G96 active the number of revs can be calculated according to the work diameter, using this formula:

$$n = \frac{V_t \times 1000}{\pi \times D}$$

Dove:

$$\left\{ \begin{array}{ll} V_t & \Rightarrow \text{cutting speed [m/min]} \\ \pi & \Rightarrow 3.14 \\ D & \Rightarrow \text{work diameter} \\ n & \Rightarrow \text{rpm} \\ 1000 & \Rightarrow \text{m to mm conversion} \end{array} \right.$$

To calculate the number of revs of machining performed at 150 m/min. on a diameter of 40:

$$\left. \begin{array}{ll} V_t & = 150 \text{ [m/min.]} \\ \pi & = 3.14 \\ D & = 40 \text{ mm} \\ n & = ? \text{ rpm.} \end{array} \right\} \quad n = \frac{150 \times 1000}{3.14 \times 40} = 1194$$

A block containing G96 is programmed:

N4

N5 **G96 S150 M4**

N6

Where:

- **G96** => Spindle speed set Vt [m/min]
- **S150** => Cutting speed Vt [m/min]
- **M4** => Spindle direction of rotation

3.10 “G92” SPINDLE REVOLUTION LIMITATIONS

Using the constant cutting speed (function **G96**) it is often necessary for technical reasons and safety (type of collet chuck, size of workpiece, unbalancing, etc.), to set a limit to the spindle maximum rpm. For example when facing or parting off, up to the centre of the workpiece the spindle speed tends to reach an infinite value. Programming “**G92 S2500**” the spindle rotates with a constant cutting speed without, however, exceeding the threshold of 2500 rpm.

Example:

N2

N3 T404

N4 **G92 S2000 ; spindle revolutions limited to a maximum of 2000**

N5 G96 S150 M4

N6 G0 X100 Z3 M8

N7

The limit set by G92 remains active until it is modified by a new setting of the same function, or it can be deactivated by programming “**G92 S0**”.

Programming G97 (fixed revs) the spindle speed limit set with G92 active is deactivated, if there is a new programming of G96 the spindle speed limit becomes active again.

At power on, if no value is specified for G92 S the spindle rotation speed will not be limited.

3.11 “G33” THREAD CUTTING MOVEMENTS

Function G33 is used for separate thread cutting movements.

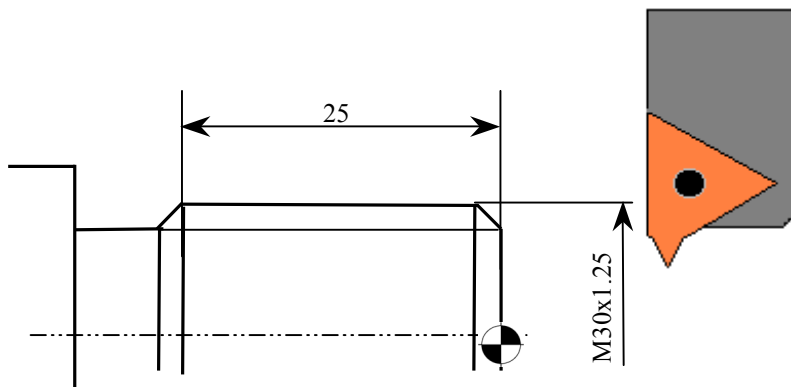
In fact, G33 differs from G1 since the tool starts the working movement only when the control receives the “spindle in position” signal from the encoder, which allows the tool to work in synchronisation with the spindle (for this reason the NC offers the possibility to cut workpieces already threaded several times, obviously without changing the gripping position).

The block with G33 may contain these instructions:

G33 final point (X or Z) feed (F) starting angle (Q)

The starting angle of the thread cutting can be programmed with address Q from 0° to 360000° (thousandths value). With the programming of a thread cutting starting angle it is possible to machine multi-start threads without moving the starting point along axis Z. If no starting angle is programmed, the NC assumes an angle of 0° as the starting value .

When machining threads, the axis and spindle trimmers are “frozen” at 100% of the programmed speed.



Example:

N1 T1 (Thread cutting)

N2 G97 S1300 M3

N3 G0 X29.5 Z5 M8

N4 G33 Z-26 F1.25

N5 G0 X32

N6 Z5

N7 X29.2

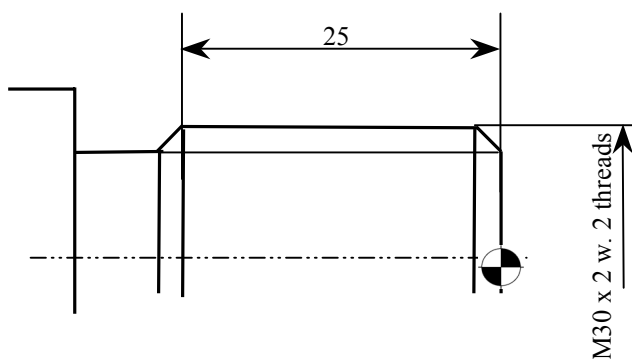
N8 G33 Z-26 F1.25

N9 G0 X32

N10 Z5

N11

Example of multi-thread machining :



N1 T1 (Thread cutting)

N2 G97 S1300 M3

N3 G0 X29.5 Z10 M8

N4 **G33 Z-26 F4 Q0**

N5 G0 X32

N6 Z10

N7 X29.5

N8 **G33 Z-26 F4 Q180000**

N9 G0 X32

N10 Z10

N11 X29.2

N12 **G33 Z-26 F4 Q0**

N13 G0 X32

N14 Z10

N15 X29.2

N16 **G33 Z-26 F4 Q180000**

N17 G0 X32

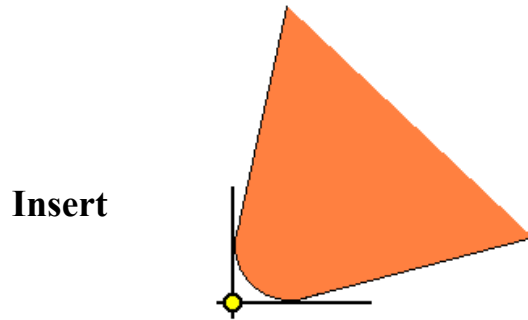
N18 Z10

N19

N20

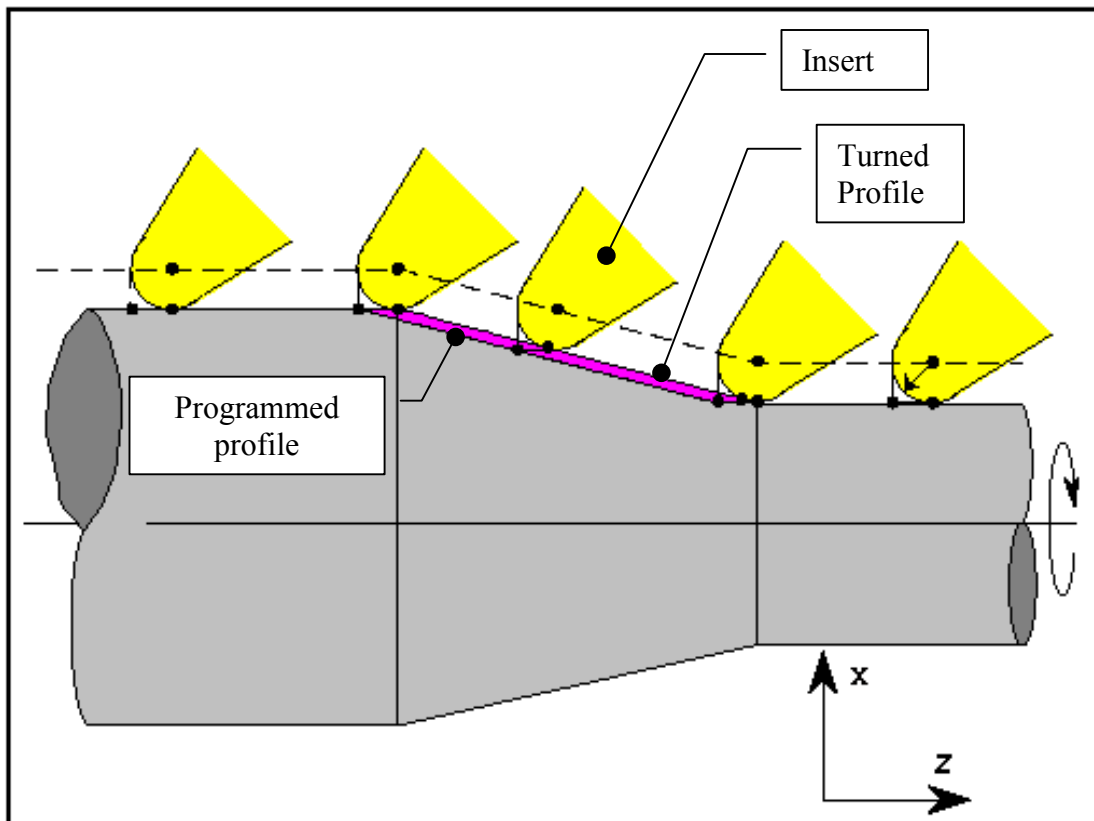
3.12 “G41”-“G42”- “G40” TOOL RADIUS OFFSET

All inserts for turning have the cutter edge rounded to a pre-defined radius, specified by the insert manufacturer (e.g. 0.4; 0.8; 1.2 etc.). With the tool measurement a point is determined for movements that is not on the insert profile, but is the intersection of the horizontal and vertical lines tangent to the insert radius, as can be seen in the figure that follows.



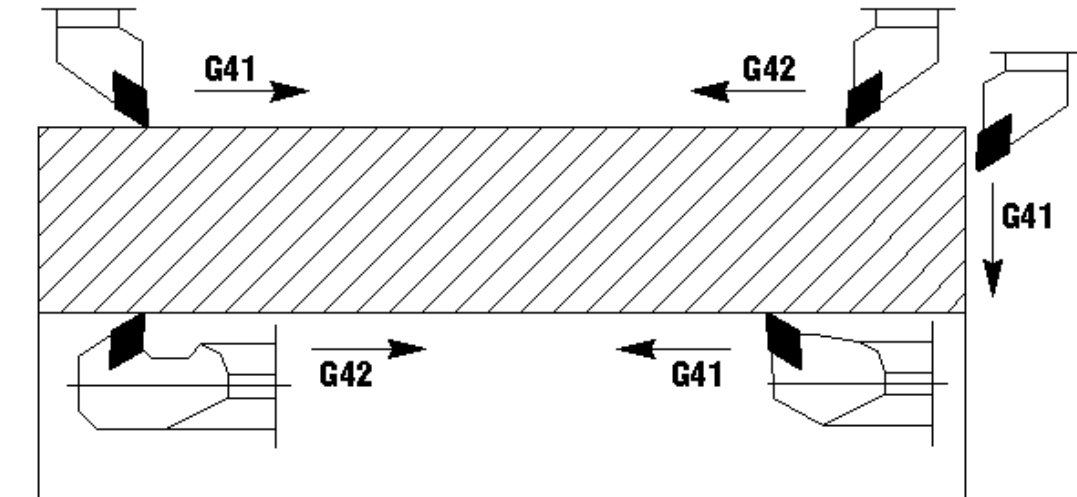
This difference has no influence when turning cylindrical parts at 90° but causes an error when machining conical and /or spherical parts, creating a profile that is not the same as that programmed. The value of this error is proportional to the insert radius and assumes the maximum value in the case of a conical profile at 45°:

$$\text{Error} = 0.412 \times \text{Insert radius}$$



To use the Tool Radius Offset therefore means to enable 3 functions from the program:

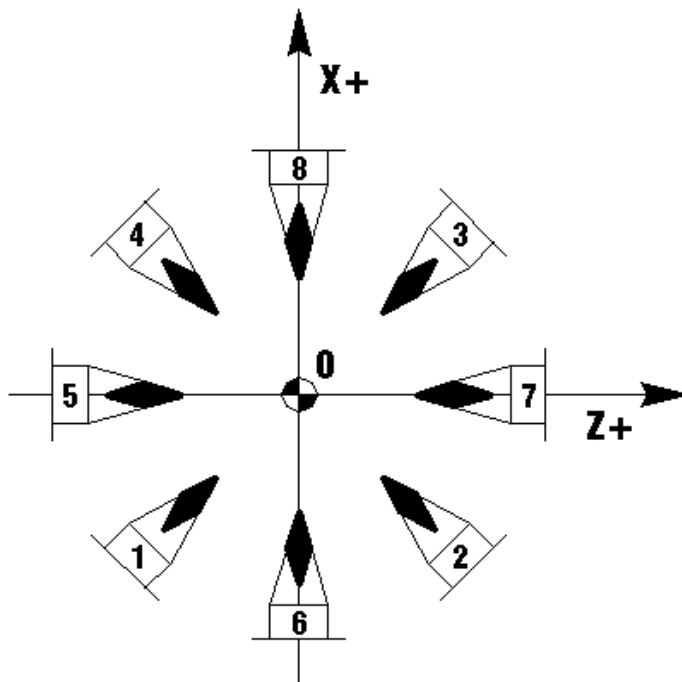
- G41** ➔ Activate the Tool Radius Offset for a **PIECE ON THE RIGHT** as to the tool direction.
- G42** ➔ Activate the Tool Radius Offset for a **PIECE ON THE LEFT** as to the tool direction.
- G40** ➔ Deactivate the tool radius offset.



The Tool Radius Offset is usually only used in the finishing stages to obtain the correct profile. In fact, this programming makes it possible to define exactly the profile specified on the drawing allowing the control to automatically offset the errors caused by the insert position and radius. To work with offset the instructions must be entered in the program to activate and deactivate the function and to supply the control with the information regarding the insert (radius and orientation).

On machines fitted with the back spindle option the activation functions (G41 / G42) and deactivation functions (G40), are applied as described in the previous diagram.

When using the Tool Radius Offset it is also necessary to enter the value of the insert radius (R) and tool orientation (T) in the tool table. The radius value is supplied by the insert manufacturer. For the tool orientation see the figure below.

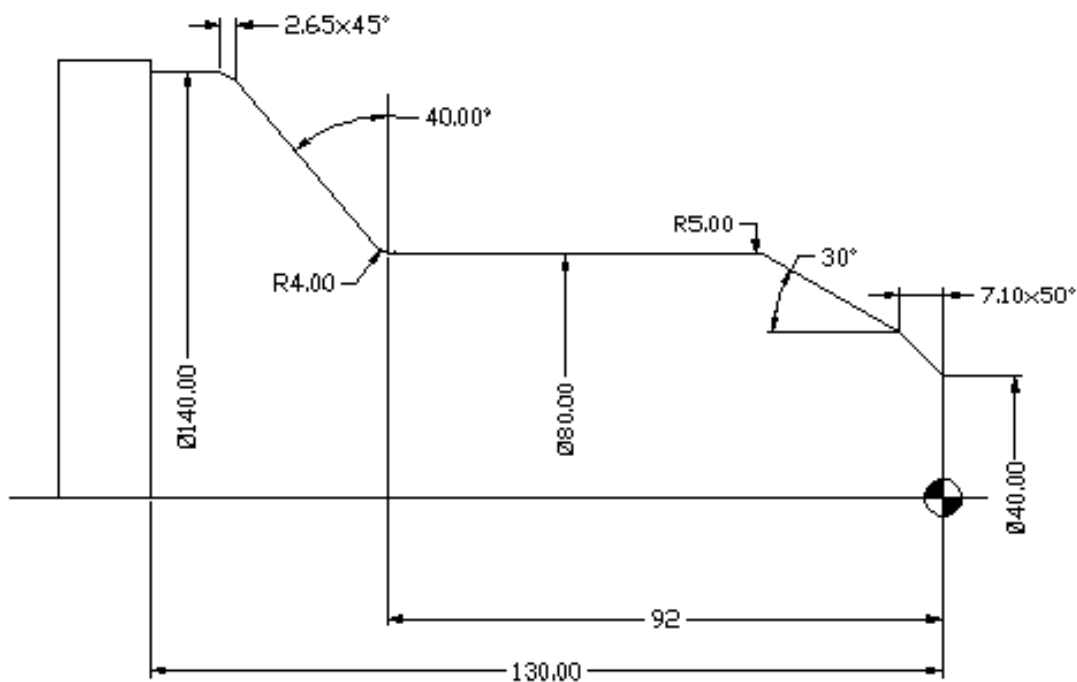


To make it simpler we can say that all the external left tools will have orientation T3 whereas all the internal left tools will have orientation T2.

When assigning a tool orientation the insert geometry is not important.

At power on, after the **RESET** key has been pressed, or after function M30, **G40** is automatically activated, furthermore it is not possible to activate and deactivate the radius offset inserting the instruction (G42 or G41) in a block with a circular interpolation movement.

Example of workpiece finishing with a tool radius 0.8:



N1 T101 (FINISHING)

N2 G92 S3000

N3 G96 S180 M4

N4 G0 X-2 Z3 M8

N5 **G42 (Activation of Tool Radius Offset)**

N6 G1 X0 Z0 F0.25

N7 X40 Z0

N8 Z-7.1 ,A130

N9 X80 ,A150 R5

N10 Z-92 R4

N11 X140 ,A130 ,C2.65

N12 Z-130

N13 X160

N14 **G40 (Deactivation of tool Radius Offset)**

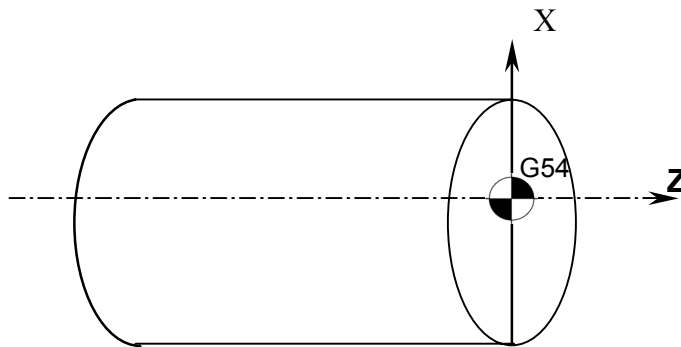
N15 G0 X200 Z200 M5

N16 M30

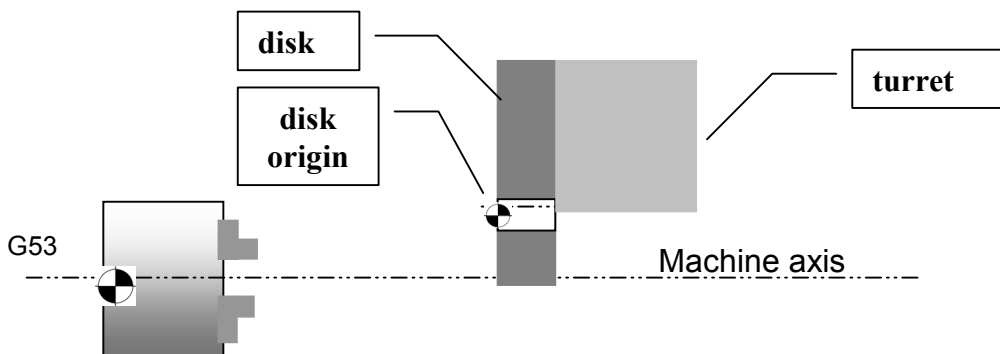
Note: enter radius (R) 0.8 and tool orientation (T) 3 in the correctors table

3.13 “G54 / G59” WORKPIECE ORIGINS

To be able to refer the tool movements to a fixed point on the workpiece to be machined. By means of a certain operation procedure one or more fixed points are defined that allow the operator to have a reference for the movements to be entered in the work program. These points are called “**WORKPIECE ORIGINS**” (G54, G55, ...G59). Usually these points are on the front of the workpiece near the spindle rotating axis.



There is also a fixed reference point that cannot be modified, created by the machine manufacturer. This point is called **MACHINE ORIGIN** (G53).



This point is used as a primary reference point and as a consequence to define the Workpiece Origins. In other words, the Workpiece Origins are found as the distance between the fixed point of the machine (G53) and our reference point on the workpiece. There is, in fact, a table where the distances from the Machine Origin are entered for each Workpiece Origin. In the work program it is sufficient to enter the call-up for the required origin to make it active (example: G54) without any value.

When programming, movements in relation to the machine origin G53 are only allowed in rapid traverse (with G0 movements).

Origin G53 cannot be written alone in the block. It must always be coupled to X or Z co-ordinates which identify the movement referred to the machine zero. This movement will always be carried out in rapid traverse.

In the case of a more “traditional” use of the machine origin it is recommended to use an origin that can be modified (e.g. G59) having X0 Z0 as value in the table

Example:

N2

N3 T101

N4 **G54 ; Workpiece origin activation**

N5 G92 S2000

N6 G96 S150 M4

N7 G0 X.... Z.... M8

N8

For the operating procedure of “Origin Measurement” and “Origin Modification”, see Chapter 15 of the Concise Guide for Operator.

NOTE.

- At power on the control automatically activates origin G54.
- In the program the storable origin (G54–G59) is called up but its value (X,Z,B,C,A) is to be entered directly in the origins table.

3.14 “G52” ORIGIN TRANSFER BY PROGRAM

An alternative to the origin transfer by table is the direct origin transfer by program using instruction G52.

With the G52 function it is possible to move the reference point by program (e.g.: G54, G55 etc.).

G52 operates in absolute mode in relation to the last workpiece origin selected, with the movement values inserted in the characters of address **X** and/or **Z** (e.g.: G52 X5 Z-10).

To cancel the origin transfer by program there are three possibilities :

machine reset

end of program instruction M30

instruction G52 X0 Z0 written in the program (procedure usually used).

Other functions cannot be inserted in the block where instruction G52 is programmed.

Example:

N2

N3 G54

N4

N5 **G52 Z-10** **Absolute origin transfer**

N6

N7

N8 **G52 Z0** **Cancel origin transfer**

N9

NOTE. If other storable origins (G54 – G59) are programmed with the G52 function active, the NC transfers from the value programmed in G52 to the new origin activated.

It is not possible to move the active origin in incremental mode using the G52 instruction. To get round this problem, the G52 function can be repeated several times with different values

Example:

N1 G54

N2

N3 G52 Z-10 (active origin moved by 10 mm)

N4

N5 G52 Z-20

N6

N7 G52 Z-30

N8

N9 G52 Z0 (active origin transfer cancelled)

3.15 “M134 / M135” PRECISE STOP

The tool passage from a block to another may happen in two ways:

- in execution point to point
- in continuous execution

These two ways of passage from a block to another can be enabled by two functions M, which are:

M134 execution point to point with deceleration at end of block.

With this function axes between the blocks execute a deceleration to reach the quote and then restart.

In this way you'll obtain a “precise” profile with live angles.

M135 Execution in continuous without deceleration at end of block.

With this function axes between a block and another don't decelerate and so, if feed is elevated, you have an error with rounding of edge.

This function is automatically active at power on.

We advise the use of function M134 to work profiles where a precise tolerance even on chamfers, cones and fitting is required.

When programmed this function is disabled by function M135, with the reset or with a stop program (M0, M1 or M30).

We advice to disable function M134 before executing a movement in rapid (GO).

3.16 LIST OF MAIN “G” PREPARATORY FUNCTIONS

The list below indicates the main **G** preparatory functions used to program the FANUC numeric control.

- G0** ⇨ rapid axis linear movement.
- G1** ⇨ axis linear movement in work mode.
- G2** ⇨ clockwise circular interpolation.
- G3** ⇨ anticlockwise circular interpolation.
- G4** ⇨ stand-by.
- G10** ⇨ data entry from program.
- G11** ⇨ deletes the data entry from program mode
- G17** ⇨ selection of working surface X Y.
- G18** ⇨ selection of working surface Z X.
- G19** ⇨ selection of working surface Y Z.
- G28** ⇨ return to reference point (with axis C and axis A option).
- G33** ⇨ thread cutting movement.
- G40** ⇨ radius offset disable.
- G41** ⇨ tool radius offset with workpiece on right of profile.
- G42** ⇨ tool radius offset with workpiece on left of profile.
- G52** ⇨ absolute programmable origin transfer.
- G53** ⇨ enables transfers referring to machine origin.
- G54** ⇨ modifiable origin transfer.
- G55** ⇨ modifiable origin transfer.
- G56** ⇨ modifiable origin transfer.
- G57** ⇨ modifiable origin transfer.
- G58** ⇨ modifiable origin transfer.
- G59** ⇨ modifiable origin transfer.
- G65** ⇨ single macro instruction call up.
- G66** ⇨ modal macro-instruction call-up.
- G67** ⇨ delete modal macro-instruction call-up.
- G70** ⇨ finishing cycle.
- G71** ⇨ material removal by turning.
- G72** ⇨ material removal by facing.
- G73** ⇨ profile repetition.
- G76** ⇨ thread cutting cycle with several cuts.
- G80** ⇨ delete fixed front drilling cycle.
- G83** ⇨ fixed front drilling cycle.

-
- G84** ⇨ fixed front tapping cycle (cannot be used with rotating tools).
 - G85** ⇨ fixed cycle of frontal boring.
 - G87** ⇨ fixed side drilling cycle.
 - G89** ⇨ fixed side of lateral boring.
 - G90** ⇨ programming with absolute co-ordinates.
 - G91** ⇨ programming with incremental co-ordinates.
 - G92** ⇨ spindle speed limitation.
 - G94** ⇨ feed programming in mm/min.
 - G95** ⇨ feed programming in mm/rev..
 - G96** ⇨ constant cutting speed programming in m/min.
 - G97** ⇨ fixed revolution spindle rotation programming in rpm.
 - G107** ⇨ cylindrical interpolation.
 - G112** ⇨ polar co-ordinates interpolation
 - G113** ⇨ delete polar co-ordinates interpolation.
 - G174** ⇨ radial grooves rough machining/pre-finishing cycle.
 - G175** ⇨ radial grooves finishing cycle.
 - G176** ⇨ axial grooves rough machining/pre-finishing cycle.
 - G177** ⇨ axial grooves finishing cycle

4.0 FIXED FANUC CYCLES

Fixed cycles are functions that simplify the ISO programming.

The most commonly used fixed cycles are described below.

4.1 “G71” MATERIAL REMOVAL BY TURNING

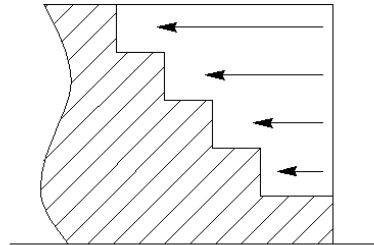
The “G71” function activates the material removal by turning cycle.

With this function the tool makes increments on axis X and turning on axis Z.

The material removal cycle in turning is always composed of two program blocks.

Example:

```
N17 .....  
N18 G0 X.. Z..  
N19 G71 U... R...  
N20 G71 P... Q... U... W... F...  
N21 G0/G1 X... Z...  
N22 ...  
N23 ... description of finished profile  
N24 ...
```

G71

Where:

- X => Start cycle co-ordinate along axis X
- Z => Start cycle co-ordinate along axis Z

1st BLOCK OF G71

- U => Depth of radial cut without sign.
- R => Tool separation in return path at 45° value without sign

2nd BLOCK OF G71

- P => Number of block where the rough machining profile starts
- Q => Number of block where rough machining profile finishes

- U => Diametric machining allowance on axis X value indicated with sign
- W => Machining allowance on axis Z value indicated with sign
- F => Work feed in rough machining

In rapid traverse the tool reaches the X and Z values indicated in the block before the first G71 (these values therefore determine the point where the tool will start to machine: X will be equal to the diameter of the blank workpiece, Z will be the safety distance that facilitates the cut increment).

An increment takes place that is equal to the radial value indicated in parameter U of the first G71 block (the increment can take place in rapid mode or work mode, depending on whether the profile description, block after the second G71, starts with a G0 or a G1).

The tool performs the rough machining automatically making several cuts, going from the point indicated in block P to the point indicated in block Q.

At the end of each cut the tool separates in rapid mode, by 45° by a radial value equal to that indicated in parameter R and returns in rapid mode to the Z starting point.

After all the rough machining cuts have been made, the tool performs a pre-finishing cut to leave even machining allowances (parameters U and W indicated with sign), and returns in rapid traverse to the starting point. Value U (that determines the diametrical machining allowance along axis X) will be positive for external machining and negative for internal machining. Parameter W (that determines the machining allowance along axis Z) will be positive for machining from the tailstock toward the spindle and negative for machining from the spindle toward the tailstock or for machining on the back spindle (on machines with this option installed)

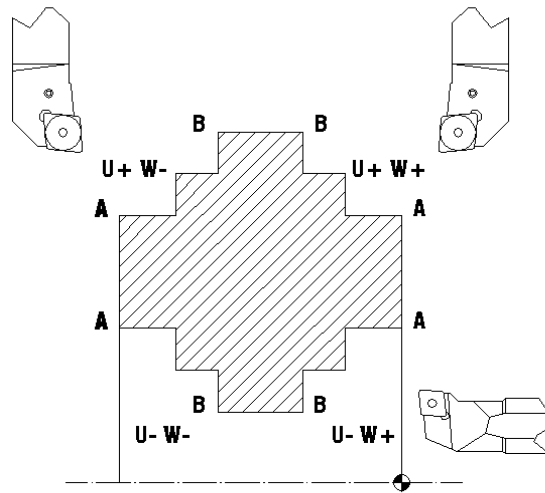
If the pre-finishing cut is not required, just program the block after the second G71, (block that starts the finished profile) to contain both X and Z.

When running the cycle the tool works with the feed programmed in parameter F of the G71 cycle, any feeds programmed in the profile description blocks are only activated during the finishing operation (see G70 cycle further on).

NOTE. The G71 rough machining cycle does not use the tool radius offset (G41, G42, G40) which can, of course, be activated in finishing (G70 cycle).

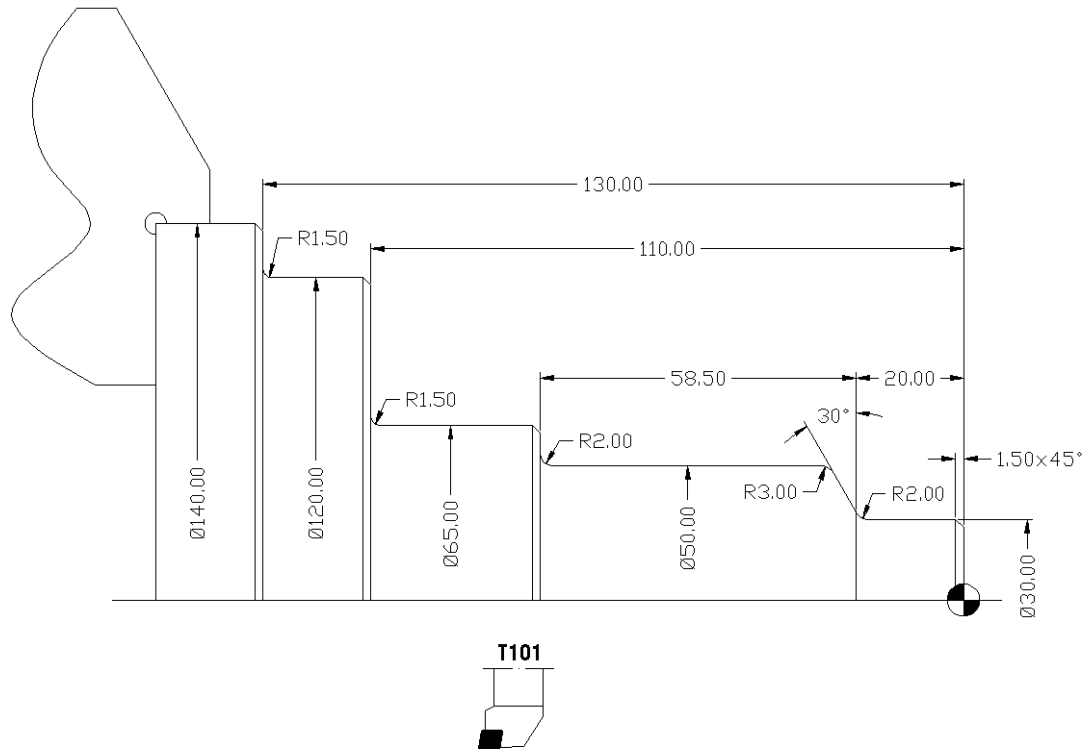
The finished profile of the workpiece cannot be managed in a sub-program, but only within the cycle itself.

For overmetal U and W situation see the scheme below:



Example of how to use the G71 cycle:

CHAMFERS 1.5 x 45°

**O3434 (REMOVAL OF MATERIAL BY TURNING)**

N1 T101

N2 G54

N3 G92 S3000

N4 G96 S200 M4

N5 G0 X140 Z3 M8

N6 G71 U3 R1

N7 G71 P8 Q19 U0 W0 F0.35

N8 G0 X26

N9 G1 Z0

N10 X30 ,C1.5

N11 Z-20 R2

N12 X50 ,A120 R3

N13 Z-78.5 R2

N14 X65 ,C1.5

N15 Z-110 R1.5

N16 X120 ,C1.5

N17 Z-130 R1.5

N18 X140 ,C1.5

N19 Z-132

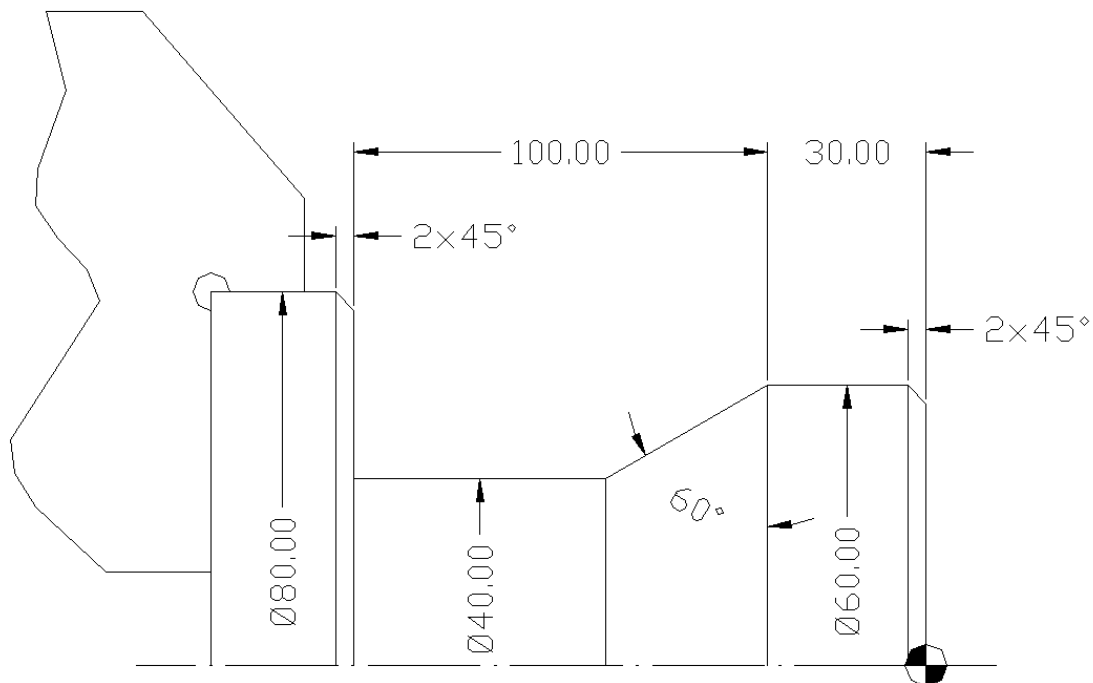
N20 G0 X200 Z200 M5

N21 M30

If in the profile there are shaded parts (decreasing profiles) proceed as follows:

- describe the shaded parts using the same functions as for monotone profiles, angles included
- the shaded parts cannot be more than 10
- the first profile description block (block after the second G71) must contain both X and Z
- remember that CNC, in machining of shaded parts, doesn't consider the tool radius compensation.

Example of how to use the G71 cycle with shaded parts :



O3435 (MATERIAL REMOVAL IN TURNING WITH SHADED PARTS)

N1 T606

N2 G54

N3 G92 S3000

N4 G96 S200 M4

N5 G0 X82 Z3 M8

N6 G71 U2 R1

N7 G71 P8 Q16 U0 W0 F0.35

N8 G0 X56 Z2

N9 G1 Z0

N10 X60 Z-2

N11 Z-30

N12 X40 ,A210

N13 Z-130

N14 X80 ,C2

N15 Z-133

N16 X83

N17 G0 X200 Z200 M5

N18 M30

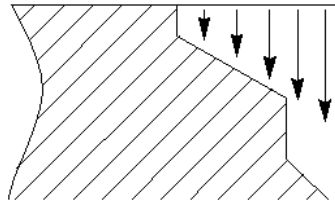
4.2 “G72” MATERIAL REMOVAL BY FACING

Function “G72” activates the material removal by facing cycle.

With this function the tool makes increments on axis Z and turning on axis X.

The material removal by facing cycle is always composed of two program blocks.

Example:

G72

```
N17 .....  
N18 G0 X.. Z..  
N19 G72 W... R...  
N20 G72 P... Q... U... W... F...  
N21 G0/G1 X... Z...  
N22 ...  
N23 ... description of finished profile  
N24 ...
```

Where:

- X => Start cycle co-ordinate along axis X
- Z => Start cycle co-ordinate along axis Z

1st BLOCK OF G72

- W => Depth of radial cut without sign.
- R => Tool separation in return path at 45° value without sign

2nd BLOCK OF G72

- P => Number of block where the rough machining profile starts
- Q => Number of block where rough machining profile finishes
- U => Diametric machining allowance on axis X value indicated with sign
- W => Machining allowance on axis Z value indicated with sign
- F => Work feed

The tool, in rapid traverse, reaches the X and Z values indicated in the block before the first G72 (these values thus determine the point where the tool will start machining: X will be equal to the rough workpiece diameter plus a small safety margin that facilitates the cut increment, Z will be 0 if the workpiece is already faced, or 1 or 2 if there is a machining allowance).

The increment will be equal to the value indicated in parameter W of the first G72 block (the increment may be in rapid mode or working mode – this depends on whether the profile description in the block after the second G72, starts with G0 or G1).

The tool makes the rough machining automatically performing a series of cuts going from one point indicated in block P up to the point indicated in block Q.

At the end of each cut the tool separates by 45°, in rapid mode, for a radial value equal to that set in parameter R and returns in rapid traverse to the Z starting point.

When all the rough machining cuts have been performed the tool makes a pre-finishing cut to leave even machining allowances (parameters U and W indicated with sign), and returns in rapid traverse to the starting point. Value U (which determines the diametrical machining allowance along axis X) will be positive for external machining and negative for internal machining, parameter W (that determines the machining allowance along axis Z) will be positive for machining from the back spindle toward the spindle, and negative for spindle machining toward the back spindle or for machining on the back spindle (on machines that have this option)

If the pre-finishing cut is not required it is sufficient to program the block after the second G72, block from which the finished profile starts, containing in it both X and Z.

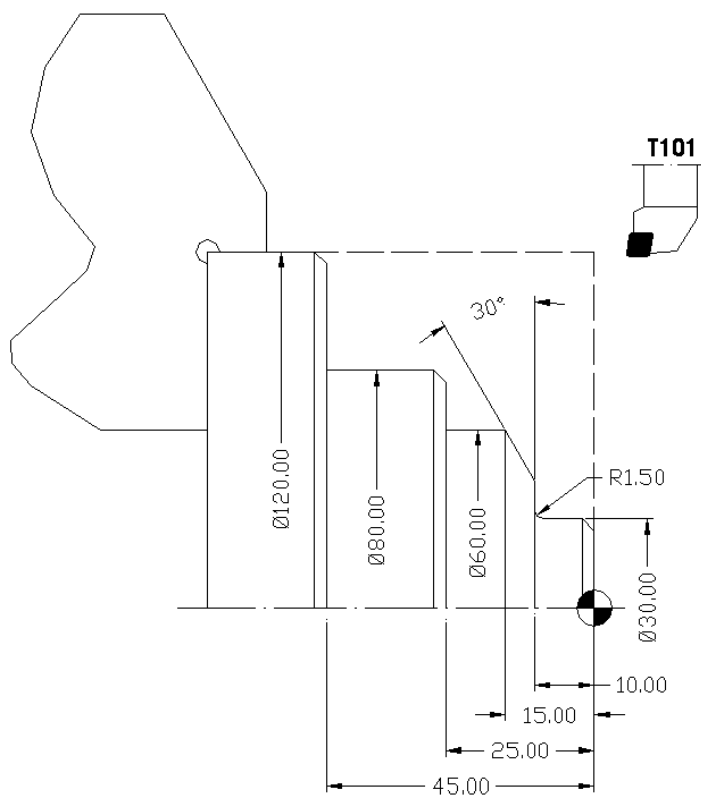
When performing the cycle, the tool works with the feed programmed in parameter F of cycle G72, any feeds set in profile description blocks are only activated during the finishing operations.

NOTE. The rough machining cycle G72 does not include the use of the tool radius offsets (G41, G42, G40) which can, of course, be activated for finishing (cycle G70).

The finished profile of the part cannot be managed in a sub-program, but only within the cycle itself.

Example of how to use cycle G72:

CHAMFERS 2 x 45°



O3435 (REMOVAL OF MATERIAL BY FACING)

N1 T101

N2 G54

N3 G92 S3000

N4 G96 S200 M4

N5 G0 X122 Z0 M8

N6 G72 W2.5 R1

N7 G72 P8 Q18 F0.35

N8 G0 Z-47

N9 G1 X120

N10 Z-45 ,C2

N11 X80

N12 Z-25 ,C1.5

N13 X60

N14 Z-15

N15 Z-10 ,A-60

N16 X30 R1.5

N17 Z0 ,C1.5

N18 X0

N19 G0 X200 Z200 M5

N20 M30

4.3 “G73” PROFILE REPETITION

The “G73” function activates the profile repetition cycle.

With this function the defined profile can be repeated several times, moving it each time by a certain distance. This cycle is most useful to work on workpieces coming from stamping, casting or a previous rough machining.

The profile repetition cycle is always composed of two program blocks.

Example:

```
N17 .....  
N18 G0 X.. Z..  
N19 G73 U... W... R...  
N20 G73 P... Q... U... W... F...  
N21 G0/G1 X... Z...  
N22 ...  
N23 ... description of finished profile  
N24 ...
```

Where:

- X => Start cycle co-ordinate along axis X
- Z => Start cycle co-ordinate along axis Z

1st BLOCK OF G73

- U => material to remove on x axe, radial value expressed with sign, (difference between rough and worked)
- W => material to remove on z axe, value expressed with sign, (difference between rough and worked)
- R => Number of profile repetitions

2nd BLOCK OF G73

- P => Number of block where the rough machining profile starts
- Q => Number of block where the rough machining profile finishes
- U => Diametrical machining allowance on axis X , value with sign
- W => Machining allowance on axis Z, value with sign
- F => Work feed

In rapid traverse the tool reaches the values of X and Z set in the block before the first G73 (thus these values determine the point where the tool will start to work).

An increment takes place which is equal to the values set in parameters U and W of the first G73 and the number of profile repetitions expressed in parameter R.

The tool makes a series of cuts going from the point set in block P up to the point set in block Q.

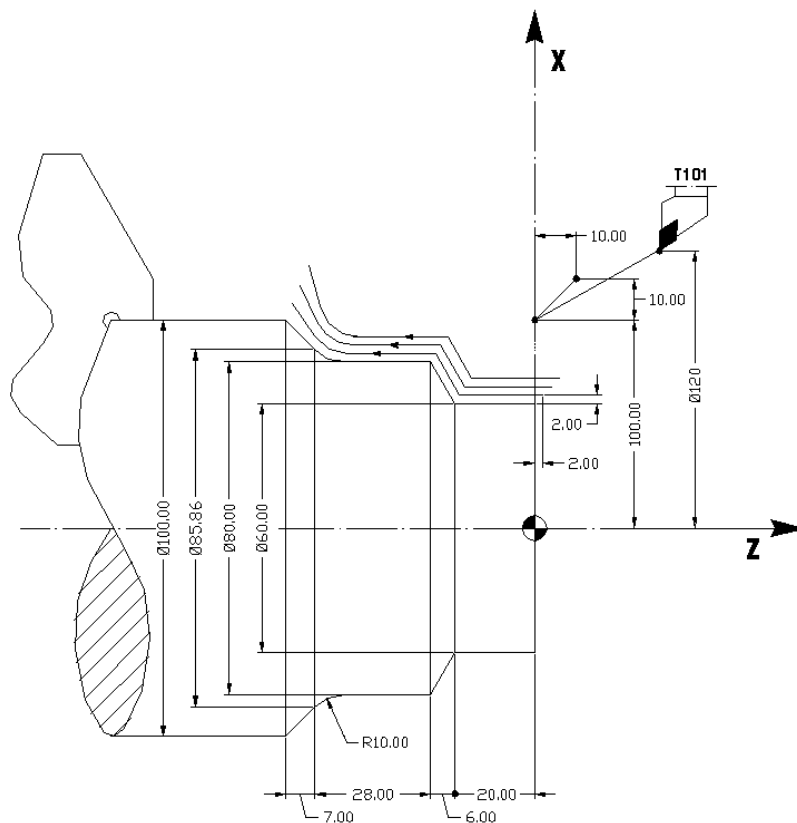
At the end of all the rough machining cuts the tool makes a pre-finishing cut to leave even machining allowances (parameters U and W , with sign), and returns in rapid traverse to the starting point. Value U (that determines the diametrical machining allowance along axis X) will be positive for external machining and negative for internal machining, parameter W (that determines the machining allowance along axis Z) will be positive for machining from the back spindle toward the spindle and negative for machining from the spindle to the back spindle or for machining on the back spindle in machines with this option)

When performing the cycle the tool works with the feed programmed in parameter F of the G73 cycle. Any feeds programmed in the profile description blocks will be activated only during finishing operations.

NOTE. the rough machining cycle G73 does not use the tool radius offsets (G41, G42, G40) which can, of course, be activated for finishing (cycle G70).

The finished profile of the part cannot be managed in a sub-program, but only inside the cycle itself.

Example of how to use cycle G73 :



O3436 (PROFILE REPETITION)

N1 T101

N2 G54

N3 G92 S3000

N4 G96 S200 M4

N5 G0 X120 Z10 M8

N6 G73 U3 W3 R4

N7 G73 P8 Q12 F0.35

N8 G0 X60 Z2

N9 G1 Z-20

N10 X80 Z-26

N11 Z-54 R10

N12 X100 Z-61

N13 G0 X200 Z200 M5

N14 M30

4.4 “G70” FINISHING CYCLE

The “G70” function activates the finishing cycle. This function can be used after the three rough machining cycles G71, G72 and G73.

The finishing cycle consists of just one block and can contain these codes:

- P => Number of first block of the profile to be finished.
- Q => Number of the last block of the profile to be finished.
- F => Finish feed.

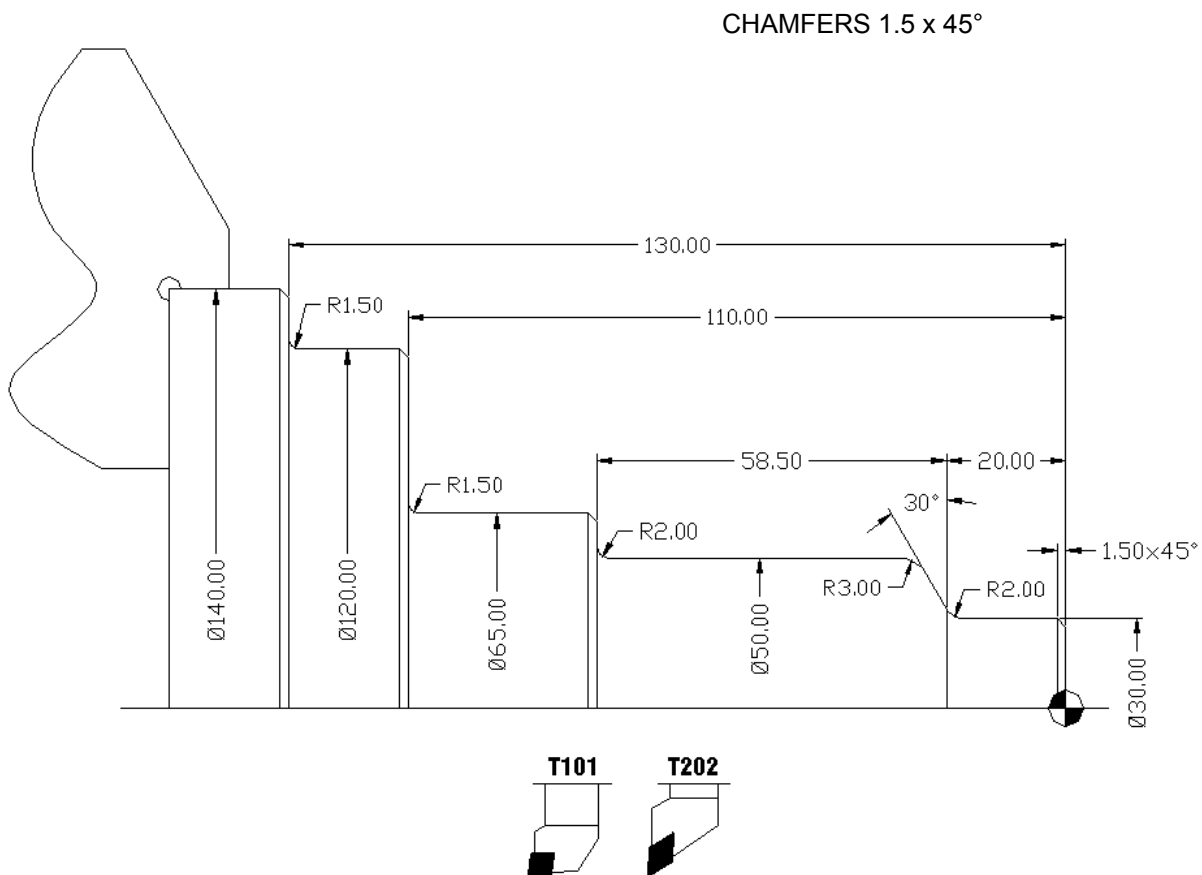
Before activating the finishing cycle G70 the tool must be positioned in the same point where the rough machining cycle G71, G72 or G73 started.

At the end of the finishing cycle the tool returns to the starting point and the NC runs the next block

There are two possibilities for the feed used in the finishing stage:

- if the whole profile is to be machined with the same feed, just specify this value inside block G70 (with parameter F)
- if various feeds are to be used on the profile, these must be specified in the profile rough machining (these feeds will be ignored in the rough machining but activated in the finishing stage)

Example of how to use cycle G70:



O3437 (PROFILE ROUGH MACHINING AND FINISHING)

N1 T101

N2 G54

N3 G92 S3000

N4 G96 S200 M4

N5 G0 X140 Z3 M8

N6 G71 U3 R1

N7 G71 P8 Q19 U0.5 W0.1 F0.35

N8 G0 X26

N9 G1 Z0

N10 X30 ,C1.5

N11 Z-20 R2

N12 X50 ,A120 R3

N13 Z-78.5 R2

N14 X65 ,C1.5

N15 Z-110 R1.5

N16 X120 ,C1.5

N17 Z-130 R1.5

N18 X140 ,C1.5

N19 Z-132

N20 G0 X200 Z200

N21 T202

N22 G54

N23 G92 S3000

N24 G96 S200 M4

N25 G0 X140 Z3 M8

N26 G70 P8 Q19 F0.15

N27 G0 X200 Z200 M5

N28 M30

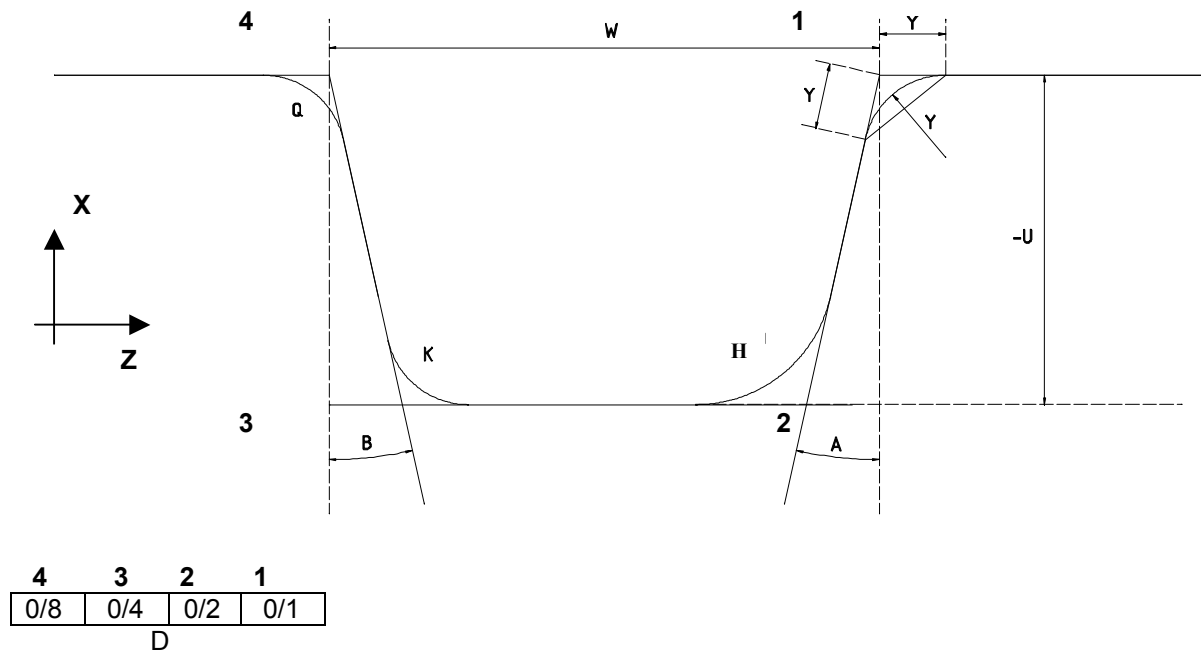
4.5 "G174" RADIAL GROOVES ROUGH MACHINING/PRE-FINISHING CYCLE

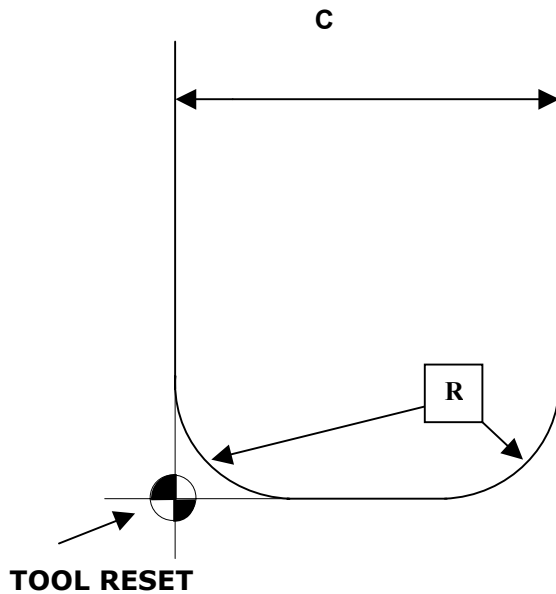
Function G174 activates the rough machining and pre-finishing cycle for grooves on outer and inner diameters, performed by a parting tool with less width at the bottom of the groove.

To run a G174 cycle the tool must be positioned with the reference edge on the start cycle point (tool reset on left edge), at a distance of a diametrical millimetre from the part to be machined.

The feed speed used is that active when the operation is called up and must be specified in a block prior to G174.

Figure 1





The insert radius of the tool used must be specified in the correctors table.

G174 must be programmed as follows:

N...G174 A.. B.. C.. U/X.. W/Z.. Y.. H.. K.. Q.. D.. (F..) (L..) (P..) (R..) (S..)

Where:

- G174 = Activates the rough machining and pre-finishing cycle for the outer and inner radial grooves.
- A.. = Angle of groove right-hand wall (in positive direction of axis Z)
- B.. = Angle of groove left-hand wall.
- These angles are always positive and have a value from 0 to 89.999 degrees. When the assigned value is 0, this means that the walls are vertical.
- C.. = Tool width , always positive value,(radius R and orientation type 3 must be specified in offset table as radius compensation is activated automatically).
- U/X.. = U indicates the depth of the groove, X indicates the value of the bottom of the groove
- specify one or the other - :
- If $U < 0$ = external groove
- If $U > 0$ = internal groove
- If $X < \text{value of starting point}$ X = external groove
- If $X > \text{value of starting point}$ X = internal groove

W/Z.. = W groove width, Z last point of groove – specify one or the other -:
If W<0 the groove machining is made from right to left of the workpiece.
If W>0 the groove machining is made from left to right of the workpiece.
If Z < the starting point value, machining is made from right to left of the workpiece (toward Z negative).
If Z > the starting point value, machining is made from left to right of the workpiece (toward Z positive).

Y*.. = Linking radius or dimension of chamfer 1 (upper external)
H*.. = Linking radius or dimension of chamfer 2 (upper internal)
K*.. = Linking radius or dimension of chamfer 3 (lower internal)
Q*.. = Linking radius or dimension of chamfer 4 (lower external)

If Y,H,K,Q, are omitted, the cycles considers them as 0.

This means that they will be eliminated from the machining (sharp edge).

D.. = Defines the type of profile (chamfer or radius) in points 1,2,3,4 (figure 1).

Bit 3	Bit 2	Bit 1	Bit 0	Binary example of number D
0/8	0/4	0/2	0/1	
8	4	2	1	

D can assume a value from 0 to 15 according to the elements (chamfers/radii) that constitute the groove and how they are arranged.

First element : may assume value 0-1 (0= Chamfer, 1= Radius)
Second element : may assume value 0-2 (0= Chamfer, 2= Radius)
Third element : may assume value 0-4 (0= Chamfer, 4= Radius)
Fourth element : may assume value 0-8 (0= Chamfer, 8= Radius)

On the basis of the sum of the elements the value is calculated for parameter D (see figure 1).

F.. = overmetal on end of groove(vertical), radial value and expressed in mm.

L.. = overmetal on groove's sides value expressed in mm.

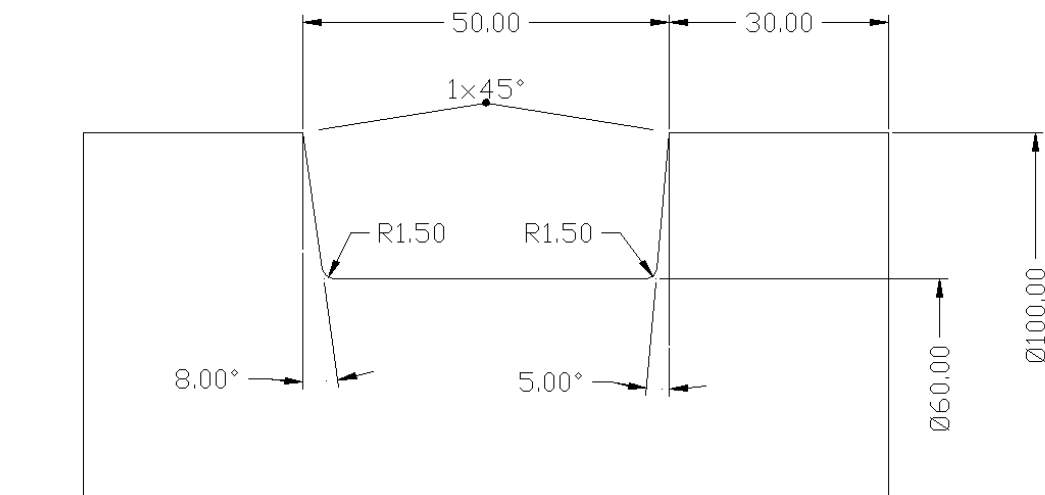
NOTE: if only one of the two variables (F or L) is specified, the other variable will be assigned the same value. If they are omitted both are considered null.

P.. = Depth of cut (it must always be more than 0, radial value expressed in mm.). The distance between one cut and another is 0.2 mm. If omitted the machining is executed in one cut.

R.. = Specifies how many grooves (cycle repetition); if omitted, default value is 1.

S.. = Specifies the centre distance for groove repetition. It can be omitted if only one groove is programmed (R=1). The value is expressed in mm and may be positive or negative.

Example of rough machining and pre-finishing of a radial groove with a tool having a width of 3 mm:



N18 T303 (TOOL FOR RADIAL GROOVES)

N19 G54

N20 G92 S1500

N21 G96 S100 M4

N22 G0 X101 Z-30 M8 F0.12

N23 G174 A5 B8 C3 X60 Z-80 Y1 Q1 H1.5 K1.5 D6

N24 G0 X200 Z100 M5

N25 M30

4.6 "G176" AXIAL GROOVES ROUGH MACHINING/PRE-FINISHING CYCLE

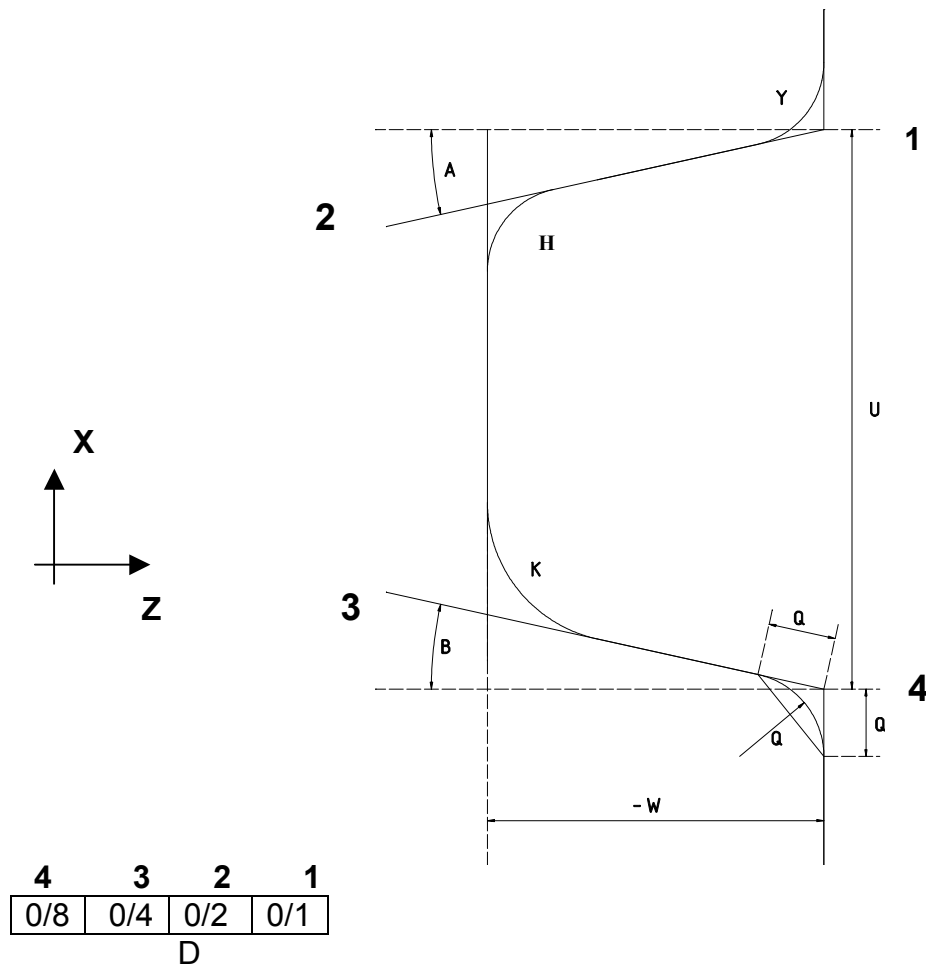
The G176 function activates the rough machining and pre-finishing cycle for axial grooves, working from the right or the left (see fig. 1) with a parting tool having a width less than the bottom of the groove

To run a G176 cycle, position the tool with the reference edge (tool reset on the bottom edge) on the start cycle point, at a distance of 0.5 millimetres from the workpiece.

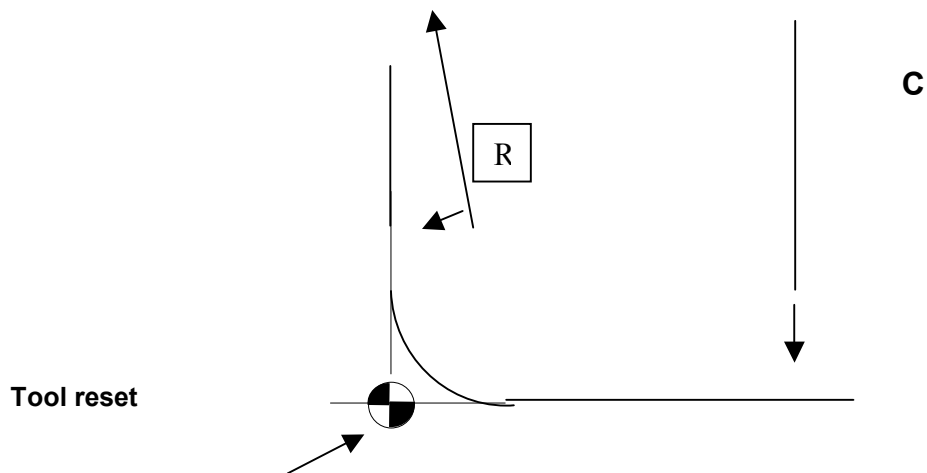
The feed speed used is that which is active when the function is called up and must be specified in a block prior to G176.

The G176 cycle disables the tool radius offset (G40)

Figure 1



Radius specified in the correctors table for the tool used.



Function G176 has to be programmed as follows:

N...G176 A.. B.. C.. U/X.. W/Z.. Y.. H.. K.. Q.. D.. (F..) (L..) (P..) (R..) (S..)

Where:

G176 = Activates the rough machining and pre-finish cycle for right and left axial grooves.

A.. = Angle of groove high wall (in axis X positive direction)
 B.. = Angle of groove low wall.
 These angles are always positive and have a value from 0 to 89.999 degrees. When the assigned value is 0, this means the walls are horizontal.

C.. = Tool width, always positive value,(r radius and orientation T3 must be specified on table offset as it's automatically activated the radius compensation).

U/X.. = U width of groove, X last point of groove – specify one or the other - :
 If $U < 0$ the groove machining takes place from top to bottom.
 If $U > 0$ the groove machining takes place from bottom to top.
 If $X <$ of the start point value, machining takes place from top to bottom of the workpiece (X negative direction).
 If $X >$ of the start point value, machining takes place from bottom to top of the workpiece (X positive direction).

W/Z.. = W indicates the depth of the groove, Z indicates the groove bottom measurement - specify one or the other - :
 If $W < 0$ = concave groove to left (Z negative direction)
 If $W > 0$ = concave groove to right(Z positive direction)
 If $Z <$ of X start point value = concave groove to left
 If $Z >$ of X start point value = concave groove to right

Y*.. = Linking radius or chamfer dimension 1 (upper external))

H*.. = Linking radius or chamfer dimension 2 (upper internal)

K*.. = Linking radius or chamfer dimension 3 (lower internal)

Q*.. = Linking radius or chamfer dimension 4 (lower external)

If Y,H,K,Q, are omitted, the cycle considers them as 0.

This means that they will be ignored when machining (sharp edge).

D.. = Defines the type of profile (if chamfer or radius) in points 1,2,3,4 (figure 1).

Bit 3	Bit 2	Bit 1	Bit 0	
0/8	0/4	0/2	0/1	Binary example of number D
8	4	2	1	

D can have a value from 0 to 15 according to the elements (chamfers/radii) that constitute the groove, and their arrangement.

First element : can have a value 0-1 (0= Chamfer, 1= Radius)
Second element : can have a value 0-2 (0= Chamfer, 2= Radius)
Third element : can have a value 0-4 (0= Chamfer, 4= Radius)
Fourth element : can have a value 0-8 (0= Chamfer, 8= Radius)

Based on the sum of the elements the value of parameter D is calculated D (see figure 1).

F.. = Overmetal on bottom (vertical), radial value expressed in mm.

L.. = Overmetal on sides (horizontal), value expressed in mm.

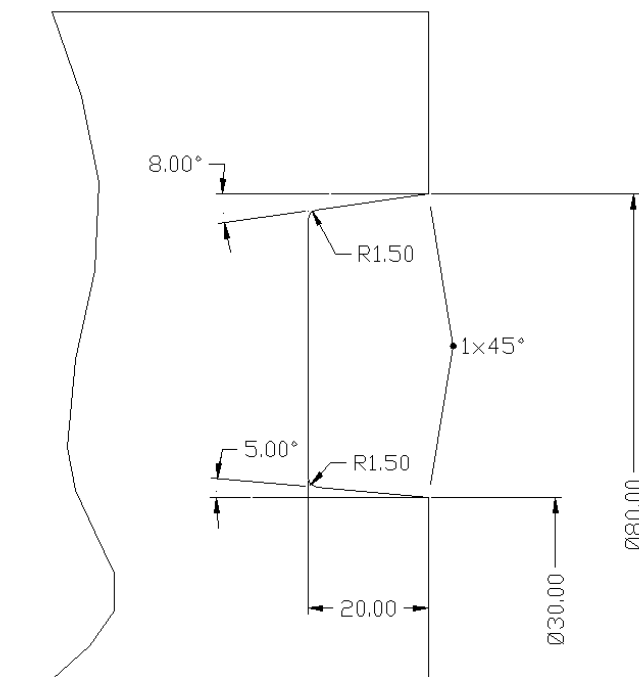
NOTE: if only one of the variables is specified (F or L) , the other variable will be assigned the same value. If they are omitted both will be considered null.

P.. = Depth of cut (must always be greater than 0). Value expressed in mm.
The distance between one block and another is equal to 0.2 mm.
If this value is omitted the groove is executed in one cut.

R.. = Specifies the number of grooves (cycle repetition); if omitted, default value is 1.

S.. = Specifies the centre distance for groove repetition. It can be omitted if only one groove is programmed (R=1). The value is radial and expressed in mm and can be positive or negative.

Example of rough machining and pre-finish of an axial groove using a tool 3 mm wide:



N18 T909 (TOOL FOR AXIAL GROOVES)

N19 G54

N20 G92 S1500

N21 G96 S100 M4

N22 G0 X30 Z0.5 M8 F0.12

N23 G176 A5 B8 C3 X80 Z-20 Y1 Q1 H1.5 K1.5 D6

N24 G0 X200 Z100 M5

N25 M30

4.7 “G175” / “G177” FINISHING CYCLE FOR RADIAL/AXIAL GROOVES

Functions G175 and G177 activate the finishing cycle for radial grooves (on outer and inner diameters) and axial grooves (cut from right to left of the workpiece).

Only function G175 is described here; the description is also valid for cycle G177 (for which the rough machining cycle is G176).

The tool position and reset follows the rules already described for rough machining cycle G174 which should be consulted for further details.

The feed speed used is that which is active when the function is called up, and must be specified in a block prior to G175.

The G175 cycle disables the tool radius offset (G40).

The parameters used are the same as for cycle G174, except for parameters F, L, P which are not used.

To activate the finishing cycle there are two syntaxes that can be used:

N...G175 A.. B.. C.. U/X.. W/Z.. Y.. H.. K.. Q.. D.. (R..) (S..)

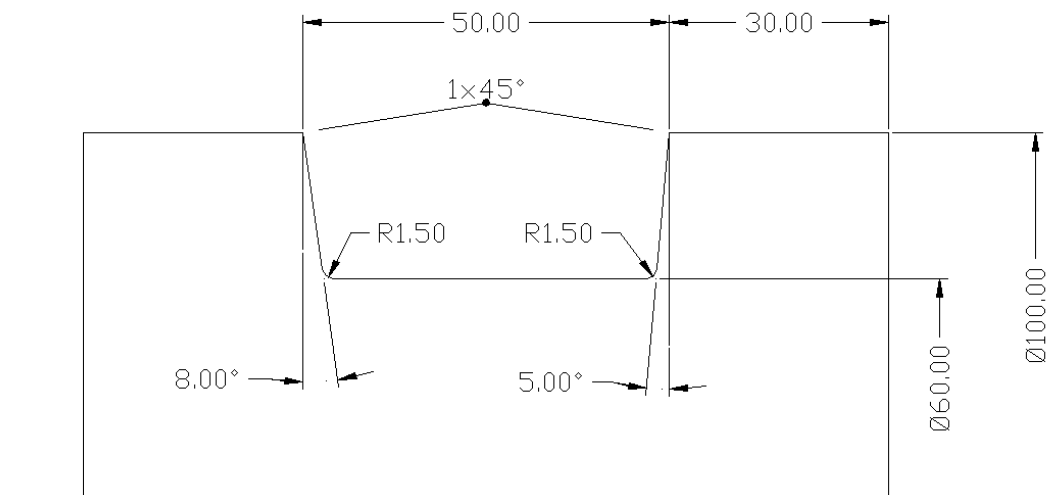
In this case all the parameters are specified (see cycle G174).

N...G175 (C..)

In this second case all the parameters indicated in the last rough machining cycle that was run are used except, (if specified) parameter C (tool width).

In both cases the corrector and the radius of the cutter used by the cycle are those active when G175 is run.

Example of rough machining and finish on a radial groove with a tool 3 mm wide:



N18 T303 (TOOL FOR RADIAL GROOVES)

N19 G54

N20 G92 S1500

N21 G96 S100 M4

N22 G0 X101 Z-30 M8 F0.12

N23 G174 A5 B8 C3 X60 Z-80 Y1 Q1 H1.5 K1.5 D6 F0.4 L0.1

N24 G175

N24 G0 X200 Z100 M5

N25 M30

If the grooves cycle is not programmed correctly, the following alarms may be generated:

All 3000:	Parameter X or U missing:	The value for parameter X or U has been omitted.
All 3001:	Parameter Z or W missing:	The value for parameter Z or W has been omitted.
All 3002:	Parameter C not correct:	The value for parameter C has been omitted or the value is less than or equal to 0.
All 3003:	Wrong tool data:	The tool radius programmed is larger than the tool width divided by 2.
*All 3004:	Slot geometry error:	The slot geometry is not correct.
All 3005:	Lower right radius error: radius (only for external grooves).	The tool radius programmed is larger than the lower right
All 3006:	Upper right radius error: right radius (only for internal grooves).	The tool radius programmed is larger than the upper
All 3007:	Upper left radius error: radius (right groove bore).	The tool radius programmed is larger than the upper left
All 3008:	Upper right radius error:: radius (left groove bore).	The tool radius programmed is larger than the upper right
All 3009:	Lower left radius error: radius (only for external grooves).	The tool radius programmed is larger than the lower left
All 3010:	Upper left radius error: radius (only for internal grooves).	The tool radius programmed is larger than the upper left
All 3011:	Lower left radius error:: radius (right groove bore)	The tool radius programmed is larger than the lower left
All 3008:	Upper right radius error:: radius (left groove bore).	The tool radius programmed is larger than the upper right
All 3012:	Lower right radius error: radius (left groove bore).	The tool radius programmed is larger than the lower right
*All 3013:	Lower right chamfer error: tool radius (only external grooves).	Lower right chamfer too small in relation to programmed
*All 3014:	Upper right chamfer error: tool radius (only internal grooves).	Upper right chamfer too small in relation to programmed

-
- *All 3015:** Upper left chamfer error: Upper left chamfer too small in relation to programmed tool radius (right groove bore).
- *All 3016:** Upper right chamfer error: Upper right chamfer too small in relation to programmed tool radius (left groove bore).
- *All 3017:** Lower left chamfer error: Lower left chamfer too small in relation to programmed tool radius (only external grooves).
- *All 3018:** Upper left chamfer error: Upper left chamfer too small in relation to programmed tool radius (only internal grooves).
- *All 3019:** Lower left chamfer error: Lower left chamfer too small in relation to programmed tool radius (right groove bore).
- *All 3020:** Lower right chamfer error: Lower right chamfer too small in relation to programmed tool radius (left groove bore).
- *All 3021:** Tool too big: Width of tool larger than the slot width or the tool base is larger than the bottom of the slot or with the tool dimensions programmed it is not possible to reach the bottom of the slot.
- All 3022:** Parameter P error: A value has been programmed for parameter P which is negative or equal to 0.
- All 3023:** Parameter F or L negative: A negative value has been programmed for parameter F or L for machining allowance.
- All 3024:** Parameter Y H K Q negative A negative value has been programmed for one or more of the parameters Y H K Q.
- All 3025:** Parameters A B error: A value has been programmed angle A and/or B which is less than 0 or greater than 90 degrees.
- All 3026:** Parameter R negative: A negative value has been programmed for the number of groove repetitions (if this value is 0 no groove is cut).
- All 3027:** Parameter S error: A number of grooves have been programmed which is greater than one, but no value (different from 0) has been set for the centre distance between grooves.

* These errors may be caused by an excessive machining allowance value in relation to the slot dimensions.

4.8 “G76” THREAD CUTTING CYCLE IN SEVERAL CUTS

Function “G76” activates the thread cutting cycle in several cuts.

This function can be used for external and internal thread cutting.

The thread cutting cycle in several cuts is always composed of two program blocks.

Example:

```
N17 .....  
N18 G0 X.. Z.. .  
N19 G76 P... Q... R...  
N20 G76 X... Z... R... P... Q... F...  
N21 G0 X... Z...
```

Where:

- X => Cycle start co-ordinate along axis X (it is also the value reached by the tool in separation at the end of each cut)
- Z => Cycle start co-ordinate along axis Z

1st BLOCK OF G76

- P => Parameter P always has 6 digits (3 pairs of numbers)

1st pair : number of finishing cuts (value from 00 to 99, always two digits)

E.g. 00 no finishing cut
 01 one finishing cut
 02 two finishing cuts

2nd pair : tapered exit from thread (value from 00 to 99, always two digits)

E.g. 00 vertical exit from thread
 05 tapered exit from thread 0.5 times the cut (value equal to half the cut)
 10 tapered exit from thread 1 time the cut (value equal to cut)

3rd pair : thread cut angle (value of two digits, only 6 selections 00,29,30,55,60,80)

E.g. 00 for square thread cutting

55 for Whitworth thread cutting

60 for metric thread cutting

If threads are to be cut with an angle that differs from the 6 selections available, use value 00

In brief : P010060 (1 idle traverse, vertical exit at end of thread , thread with angle of 60°)

- Q => Minimum cut depth (in thousandths)

E.g. Q100=0.1mm.

- R => Depth of finishing cut (radial, in mm)

E.g. R0.02=0.02mm.

2nd BLOCK OF G76

- X => Bottom of thread diameter
- Z => End of thread absolute co-ordinate
- R => Taper of thread cut (radial difference between starting thread cut diameter and diameter at end of thread cutting). The value must be indicated with a sign. For cylindrical thread cutting parameter R is not indicated.

$R = (\text{START OF THREAD DIAMETER} - \text{END OF THREAD DIAMETER}) / 2$

- P => Radial height of thread (in thousandths and without sign)

The value set for P depends on the type of thread cut and can be:

P=613 for Cut for ISO metric thread

P=640 for Cut for Whitworth thread DIN 11

P=500 for Cut for square thread

Thus : P1226 (for an ISO metric thread pitch 2)

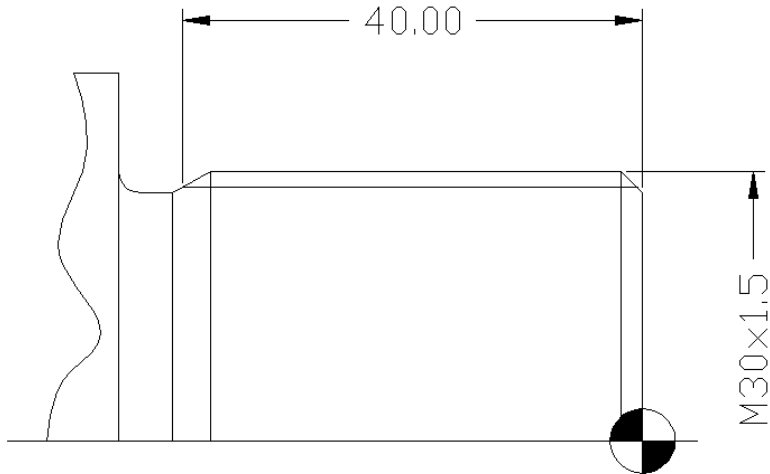
- Q => Radial depth of first cut (in thousandths)

E.g. Q250=0.25mm.

- F => Thread pitch (in mm.)

E.g. F1.5 for thread with 1.5 mm pitch.

Example of external metric thread cutting :



N17 T101 (External thread cut)

N18 G54

N19 G97 S800 M3

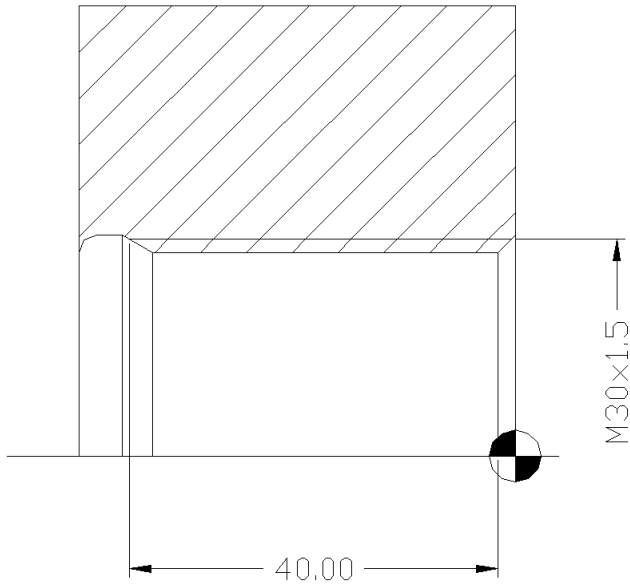
N20 G0 X32 Z6 M8

N21 G76 P010060 Q100 R0.02

N22 G76 X28.161 Z-50 P919 Q250 F1.5

N23 G0 X150 Z100

Example of internal metric thread cut :



N17 T101 (Internal thread cut)

N18 G54

N19 G97 S800 M3

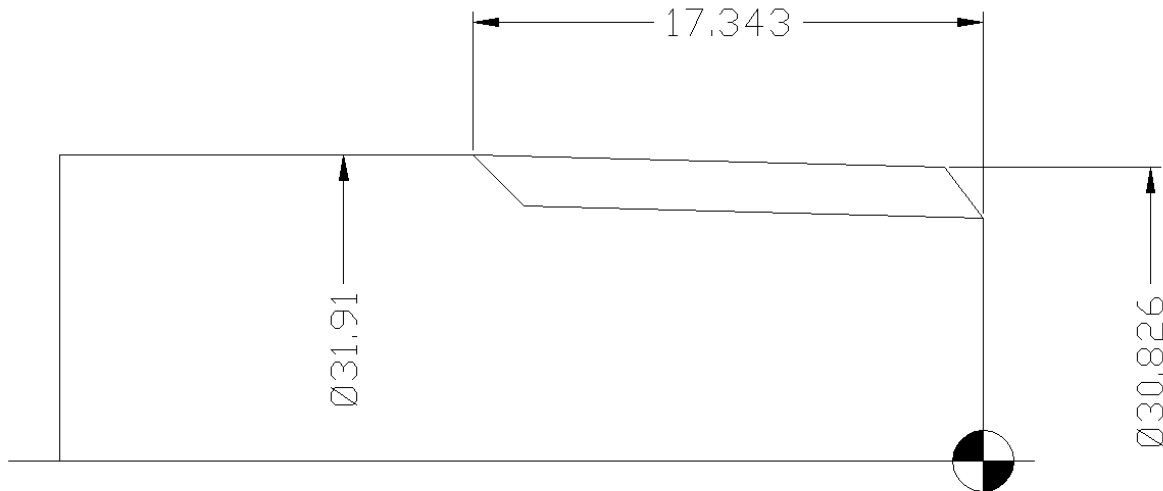
N20 G0 X25 Z6 M8

N21 G76 P010060 Q100 R0.02

N22 G76 X30 Z-40 P919 Q250 F1.5

N23 G0 X150 Z100

Example of external tapered thread cut 1" NPT (pitch 14 threads / inch) :



N17 T101 (Taper thread cut)

N18 G54

N19 G97 S800 M3

N20 G0 X33 Z6 M8

N21 G76 P010060 Q100 R0.02

N22 G76 X29.588 Z-17.343 P1161 Q250 F1.814 R-0.729

N23 G0 X150 Z100

To machine the tapered thread cut it is important to remember:

- Pitch **F = 25.4** (comparison between mm and inches) / **14** (n° threads / inch) = **1.814** mm
- **P** is calculated by multiplying the pitch by 640 (**1.814 x 640 = 1161**)
- End of thread **X** refers to the final diameter **31.91 – [(0.64 x 1.814) x 2] = 29.588**
- The starting diameter to calculate **R** is that relating to the starting **Z** (in the example Z6); in this case making the calculation with the aid of trigonometry, the result is X29.367,
- Therefore **R** Will be **(30.451-31.91):2=-0.729**

4.9 “G83” FRONT DRILLING CYCLE

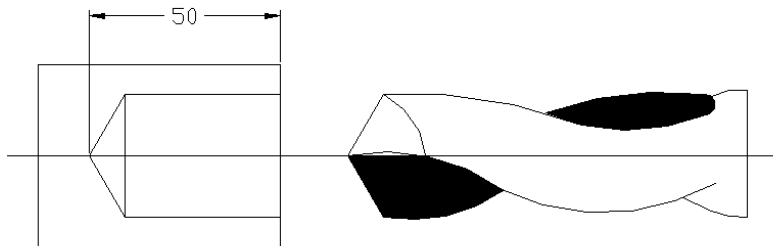
Function “G83” activates the front drilling cycle.

With this function the bit makes a series of cuts, of the required size, undercutting or breaking the chip and returning, at the end of cycle, in rapid traverse to the starting point.

The front drilling cycle can contain these codes:

- Z => End of drilling absolute value
- F => Drilling feed (in mm/rev.)
- Q => Depth of cut (in thousandths)
- P => Pause on bottom of hole (in thousandths of second)

Example:



N12 T303 (DRILLING)

N13 G54

N14 G97 S800 M3

N15 G0 X0 Z5 M8

N16 G83 Z-50 F0.12 Q1000

N17 G0 X200 Z200

Codes Q and P if not used, need not be written.

This cycle can be used to break or undercut chips according to the value of parameter 5101 bit 2 (RTR), with value 0 chip breaking, with value 1 chip undercutting, by default this bit is set to 1 therefore for chip undercutting.

It should also be remembered that parameter 5114 determines:

- with chip undercutting: the distance at which the drill is to stop in relation to the last point reached, when re-entering the hole after undercutting.
- with chip breakage: by how much the drill is to come backward between one drilling cut and the next

To cancel the drilling cycle it is necessary to program function G80 or any G function in group 01, therefore G0, G1, G2, or G3.

NOTE: On all Graziano Spa machine models the axial tool resetting (tapping drills, cutters etc.) is made only along axis Z but it is necessary to enter zero in the X location of the tool table for the tool used (See Concise Guide for Operator, paragraph 3.2)

4.10 "G84" FRONT TAPPING CYCLE

THIS CYCLE IS NOT VALID FOR TAPPING WITH MOTOR DRIVEN TOOLS

FOR TAPPING WITH MOTOR DRIVEN TOOLS SEE FUNCTIONS P9103 AND P9104

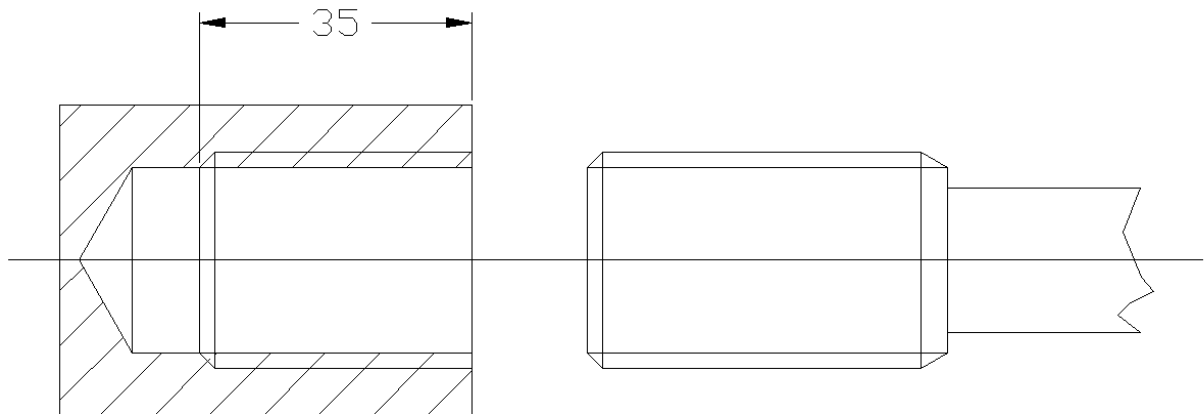
Function "G84" activates the front tapping cycle.

With this function the tapping drill enters with a feed equal to the tapping pitch, feed reduction and spindle revs to reach the end point of the tapping simultaneously, reverse spindle rotation, simultaneous acceleration of spindle and axis and return to starting point.

The front drilling cycle can contain these codes:

- Z => End of tapping absolute value
- F => Tapping pitch (in mm/rev.)

Example:



N12 T404 (TAPPING M10 x 1.5)

N13 G54

N14 G97 S300 M3

N15 G0 X0 Z5 M8

N16 G84 Z-35 F1.5

N17 G80

N18 G0 X200 Z200

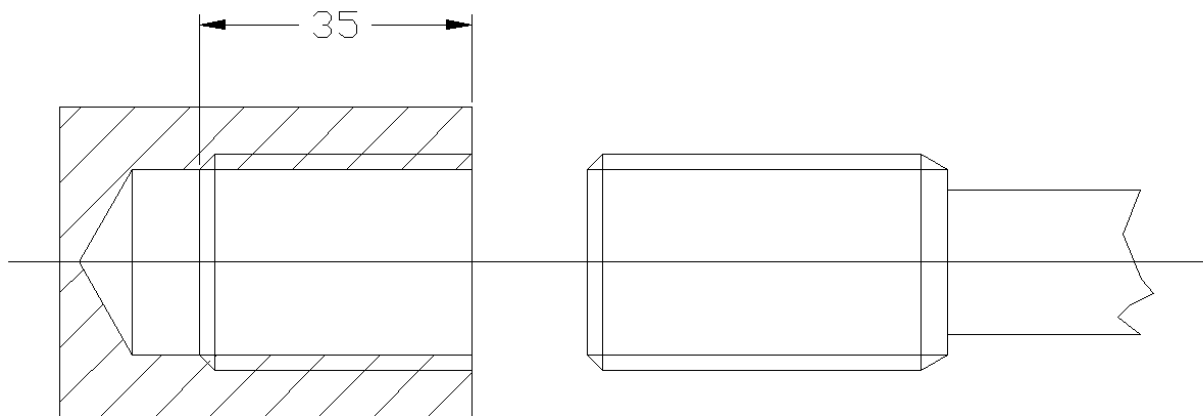
This cycle can only be used for right-hand tapping, therefore with entry in direction M3 and coming out in direction M4. If a left-hand tapping is required, two parameter values have to be changed :parameter 5112 (entry direction of rotation, usually 3) and parameter 5113 (direction of rotation coming out, usually 4) To execute a tapping on backspindle it's necessary to change the value of two parameters: parameter 5112 (rotation direction in entry normally 204) and the parameter 5113 (rotation direction in exit normally 203). At the end of this operation you need to recall the right values of the two parameters 5112 value 3 and 5113 value 4 already inserted by default.

To cancel the tapping cycle it is necessary to program function G80 or any G function in group 01, therefore G0, G1, G2, or G3.

This cycle can be used both for tapping with offset and for tapping without offset (rigid).

When using rigid tapping, function **M29** must be set in the block prior to the G84 cycle.

Example of rigid tapping:



N12 T404 (RIGID TAPPING M10 x 1.5)

N13 G54

N14 G97 S300 M3

N15 G0 X0 Z5 M8

N16 **M29 (RIGID TAPPING ACTIVATION)**

N17 G84 Z-35 F1.5

N18 G80

N19 G0 X200 Z200

To cancel the rigid tapping cycle, function G80 or any G function of group 01, therefore G0, G1, G2, or G3 must be programmed.

5.0 SUB-PROGRAMS / PARAMETRIC PROGRAMMING

Sub-programs are useful to repeat the same operation several times, using inside the program the same functions and co-ordinates already known to the operator

With parametric programming, variable values (parameters or variables #) can be attributed to the program codes instead of fixed values (numeric values). A value can be assigned to a variable through the program, from the MDI window, or inserting it in the variables table.

A variable is programmed with address # followed by a number

5.1 “M98” “M99” USE OF SUB-PROGRAMS

A program can be divided into main program and sub-programs.

Usually the NC operates under the control of the main program, but when an instruction is found that calls up a sub-program the control passes over to the sub-program. When an instruction to return to the main program is found, the control returns to the main program.

Sub-programs can be used when there are fixed repetitive sequences, simplifying the programming. A sub-program can be called up by the main program. A sub-program that has been called can, in its turn, call up another sub-program. Sub-program call-ups can be nested in up to four levels, as shown on the next page:

MAIN PROGRAM	SUB-PROGRAM	SUB-PROGRAM	SUB-PROGRAM
O1	O8001	O8002	O8003
...
...
...
...
...
M98 P8001	M98 P8002	M98 P8003	...
...
...
...
...
M30	M99	M99	M99

Level 1

Level 2

Level 3

A sub-program is a normal program that ends with function M99. The same functions can be used inside the sub-program as used in main programs (e.g. fixed cycles, geometric functions etc.)

To simplify their use, we suggest that sub-programs are given names from O8001 to O8999 (main programs go from O1 to O8000)

A sub-program is run when it is called by the main program or by another sub-program.

To call up a sub-program, write:

M98 P 0000	0000
Number	name of
Repetition	sub-program
(max 9999)	

When the number of repetitions is omitted the NC assumes a value of 1

Example : sub-program 8003 is to be repeated 6 times consecutively

M98 P68003

Instruction "M99" that closes the sub-program is used to return to the main program (or to the sub-program) in the block that immediately follows the running of the sub-program.

If it is required to return from the sub-program to a pre-defined block and not to the block that immediately follows the one in which it has been run, add the pre-defined block to M99, preceded by the letter P.

MAIN PROGRAM	SUB- PROGRAM
N10	O8003
N20	N10
N30	N20
N40	N30
N50	N40
N60	N50
M98P8003	N60
N70	N70
N80	N80
N90	N90
N100	N100 M99P80
N110 M30	M30

After the sub-program has run, the NC returns in the main program at block N80

Function M99 (which usually closes a sub-program) can be used also in the main program as an unconditioned skip (to skip always to a pre-defined block)

O1 (MAIN PROGRAM)

N10

N20

N30 /M99 P70 (USED TO OPTIONALLY SKIP THE PARTS OF THE PROGRAM FROM BLOCK 30 TO
BLOCK 70, SEE USE OF BARRED BLOCK)

N40

N50

N60

N70

N80

N90

N100 M30

Or to repeat continually a part of the program

O2 (MAIN PROGRAM)

N10

N20

N30

N40

N50

N60

N70

N80

N90 M99 (SKIP TO FIRST BLOCK AND CONTINUE REPEATING THE PROGRAM)

N100 M30

5.2 PARAMETRIC PROGRAMMING

Parametric programming uses variables, arithmetical instructions and conditioned skip instructions. In this way programs for general use can be developed, or they can be personalised for specific customer requirements.

VARIABLES

There are four types of variables:

From #1 to #33	LOCAL VARIABLES	These can only be used inside a macro and not shared with other macros. At power on the content of these macros is nil because they are volatile.
From #100 to #149	COMMON VARIABLES	These can be shared with other macros. At power on the content of these macros is nil because they are volatile
From #500 to #999	COMMON VARIABLES	These are like the variables from #100 to #149 with the difference that they are stable, they hold their content even with the machine switched off.
From #1000 to #....	SYSTEM VARIABLES	Used to read and write NC data, such as position of tool, of the axis and the tool correction values etc.

Since gennuary 2001 common variables “fixed” by the customers, for parametric programming, are those which range from #510 to #699; as the others are used by Graziano SPA for specific operations.

ARITHMETIC OPERATIONS

There are ten types of arithmetical operations available:

1 Variable definition and replacement

Example:

#101=1005

#101=#110

#101=-#112

2 addition

Example:

#101=#110+#111

3 subtraction

Example:

#101=#110-#111

4 multiplication

Example:

#101=#110*#111 or #101=#110*7

5 division

Example:

#101=#110/#111 or #101=#110/7

6 square root

Example:

#101=SQRT[#110] or #101=SQRT[5]

7 sine

Example :

#101=SIN[#110] or #101=SIN[30]

8 cosine

Example:

#101=COS[#110] or #101=COS[30]

9 tangent

Example:

#101=TAN[#110] or #101=TAN[30]

10 arc. cot.

Example:

#101=ATAN[#110]/ [#103]

CONDITIONED AND UNCONDITIONED SKIP INSTRUCTIONS

There are seven types of conditioned and unconditioned skip instructions:

1 unconditioned skip

Example:

GOTO1000 (skip to block N1000)

2 conditioned skip if the same

Example:

IF[#101 EQ #102] GOTO1000 (skip to block N1000 if parameter #101 is the same as
parameter #102, if the two parameters are different
the program passes to the next block)

3 conditioned skip if different

Example:

IF[#101 NE #102] GOTO1000 (skip to block N1000 if parameter #101 is different from parameter #102, if the two parameters are the same the program passes to the next block)

4 conditioned skip if greater than

Example:

IF[#101 GT #102] GOTO1000 (skips to block N1000 if parameter #101 is greater than parameter #102, if parameter #102 is greater or equal to parameter #101 the program continues with the next block)

5 conditioned skip if less than

Example:

IF[#101 LT #102] GOTO1000 (skips to block N1000 if parameter #101 is less than parameter #102, if parameter #102 is less than or equal to parameter #101 the program continues with the next block)

6 conditioned skip if greater than or equal to

Example:

IF[#101 GE #102] GOTO1000 (skips to block N1000 if parameter #101 is greater than or equal to parameter #102, if parameter #102 is greater than parameter #101 the program continues with the next block)

7 conditioned skip if less than or equal to

Example:

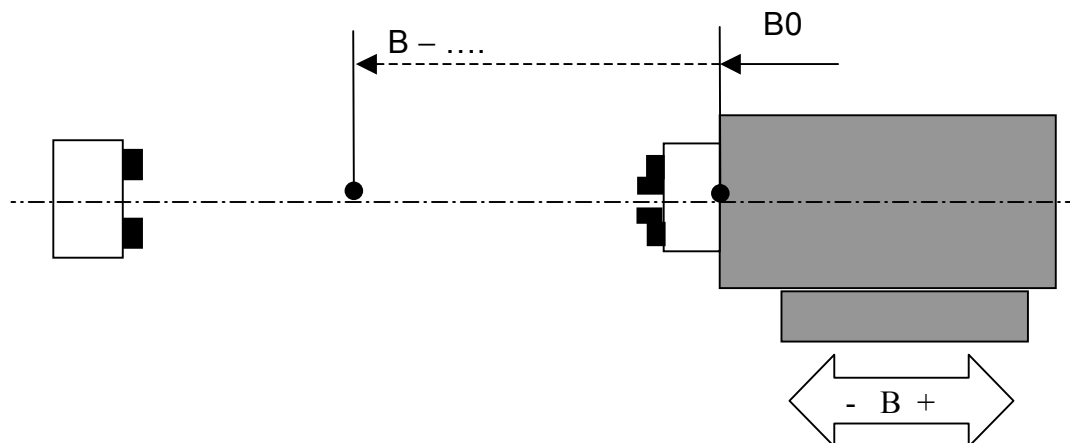
IF[#101 LE #102] GOTO1000 (skips to block N1000 if parameter #101 is less than or equal to parameter #102, if parameter #102 is less than parameter #101 the program continues with the next block)

6.0 BACK SPINDLE MACHINING

The back spindle option is an additional spindle opposite and co-axial to the main one, which makes it possible to machine on the rear part of the workpiece after taking it from the first spindle. The back spindle is useful when working on parts machined from bars, since in most cases, it is possible to obtain complete items regarding the turning operations. This option consists of a spindle mounted on a saddle that allows movement in the direction parallel to the turret axis Z.

6.1 MOST IMPORTANT ADDRESSES USED

The programming of the movements of this axis uses address B (E.g.: N54 G0 X... Z... B0). The B function can be used together with other movement co-ordinates, and in this case the movement will take place when all the axes inserted in the block simultaneously reach the programmed position.



Also the back spindle direction of rotation is controlled by special instructions: M203, M204, M205 (E.g.: N12 S1250 M203)

The block with the spindle rotation instructions is programmed as follows:

```
N24 G92 S2500          ; Revs limitation
N25 G96 S250 M204      ; Cutting speed
;
N32 G97 S1400 M203     ; Number of fixed revs
N41 G0 X... Z... M205  ; Separation from axes and back spindle stop
```

Where:

- M203 => Back spindle clockwise rotation
- M204 => Back spindle anti-clockwise rotation
- M205 => Back spindle rotation stop
- B => Back spindle axis movement co-ordinate

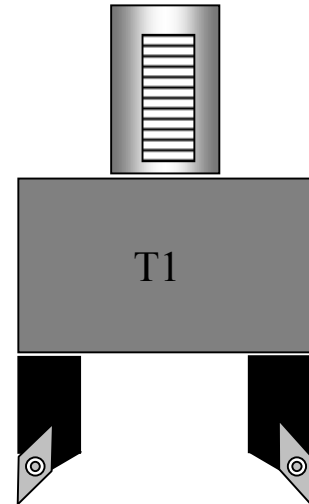
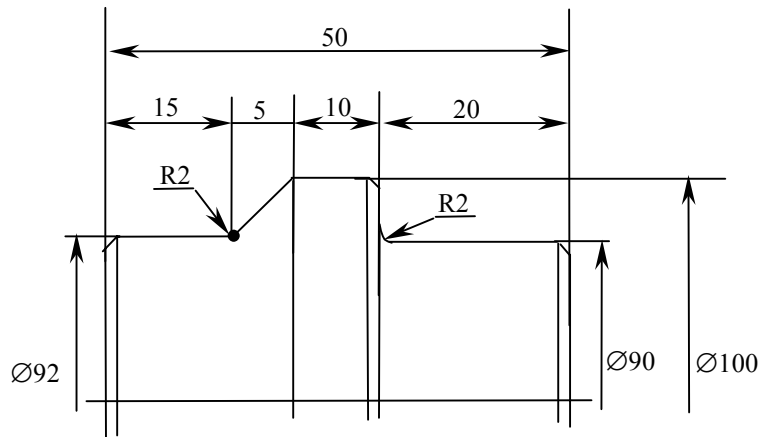
6.2 “M” AUXILIARY FUNCTIONS

The list below contains all the M functions used with the back-spindle option for many specific applications. For details on how to use these functions, consult the machine documentation.

- M10** ⇔ Active air blow jaws cleaning (back spindle rotation active with open jaws)
- M11** ⇔ air blow jaws cleaning not active (back spindle rotation not active with open jaws)
- M54** ⇔ lance to eject piece forward (option)
- M55** ⇔ lance to eject part backward (option)
- M59** ⇔ activate collet chucks or jaws washing (option)
- M60** ⇔ deactivate collet chucks or jaws washing (option)
- M70** ⇔ activate synchronising between spindle and back spindle
- M71** ⇔ activate synchronising between spindle and back spindle at a defined angle
- M72** ⇔ deactivate synchronising between spindle and back spindle
- M100** ⇔ temporary setting aside of active S
- M203** ⇔ back spindle clockwise rotation
- M204** ⇔ back spindle anti-clockwise rotation
- M205** ⇔ back spindle stop
- M213** ⇔ M203 with coolant delivery
- M214** ⇔ M204 with coolant delivery
- M219** ⇔ back spindle orientation angle is identified by S)
- M220** ⇔ back spindle brake engaged
- M221** ⇔ back spindle brake released
- M236** ⇔ axis C disabled on back spindle
- M237** ⇔ axis C enabled on back spindle
- M238** ⇔ tool reset sensor in working position
- M239** ⇔ tool reset sensor in home position
- M258** ⇔ back spindle and sensor 2 orientated in work position
- M268** ⇔ close self-centring chuck/back spindle collet chuck
- M269** ⇔ open self-centring chuck/back spindle collet chuck
- M986** ⇔ back spindle external part holder (shafts)
- M987** ⇔ back spindle internal part holder (flanges)

6.3 EXAMPLE OF MACHINING WITH BACK SPINDLE

Example of machine the part shown below with a back spindle:



```

N1 G0 B0 ; Back spindle re-positioning
N2 T101 ; Tool call-up
N3 G54 ; Origin activation
N4 G92 S2500 ; Main spindle revs limitation
N5 G96 S150 M4 ; Main spindle cutting speed
N6 G0 X103 Z0 M8
N7 G1 X-0.5 F0.25
N8 G0 X88 Z2
N9 G1 Z0
N10 X90 Z-1 F0.3
N11 Z-20 R2 ; Machining on main spindle side
N12 X100 ,C1
N13 Z-30.5
N14 G0 X200 Z200 ; Separation to change workpiece
N15 G0 B0

N16 G65 P9102 X230 V-368.5 B-376 E1000 M4 A0 Z-4 Y20 ; Workpiece change-over macro.

N17 G0 B0 ; Back spindle re-positioning
N18 T121 ; Tool call-up
N19 G55 ; Origin activation
    
```

N20 G92 S2500 ; Back spindle revs limitation
N21 G96 S150 M204 ; Back spindle cutting speed
N22 G0 X103 Z0 M8
N23 G1 X-0.5 F0.25
N24 G0 X90 Z-2
N25 G1 Z0 ; Machining on back spindle side
N26 X92 Z1 F0.3
N27 Z15 R2
N28 X100 Z20 F0.15
N29 G0 X200 Z-200 M205
N30 M30

Machining with the back spindle is exactly the same as with the main spindle ; the same ISO functions, same Fixed Cycles. Attention must be paid to the axis Z sign, which for the back spindle will be positive for machining and negative for approach and separation. We also advise the use of different origins on the first and second spindle (e.g. G54 on the main spindle and G55 on the secondary one). The passage of the part between the main spindle and the secondary spindle takes place by means of three macro which have been prepared by Graziano:

- O9100 => Workpiece change-over macro with parting off
- O9101 => Workpiece change-over macro with parting off but no extraction
- O9102 => Workpiece change-over macro without parting off

EXAMPLE OF PIECE EXCHANGE FROM MAIN SPINDLE TO BACK SPINDLE

....(Machining side main spindle)

M10 (Spindle and back spindle rotation enabling with open jaws)

M269(Confirm back spindle opening jaws)

G97 S500 M4 (Spindle rotation for synchronism)

M71 (Synchronism between spindles in speed and phase active)

G0 B- ...(back spindle for piece exchange positioning)

M268 (closing back spindle jaws)

M69 (Opening spindle jaws)

G0 B0 (Positioning back spindle for further machinings)

M72 (Synchronism between spindles not enabled)

M11 (Spindle and back spindle rotation with open jaws not enabled)

....(Machining side back spindle)

EXAMPLE OF PIECE EXCHANGE WITH COUPLE REDUCTION

....(Machining side spindle)

M10 (Spindle and back spindle rotation with open jaws enabling)

M269 (Confirm opening jaws of back spindle)

G97 S500 M4 (Spindle rotation for synchronism)

M71 (Synchronism between spindles in speed and phase enabling)

G0 B- (Back spindle positioning at one mm from the quote of piece exchange)

G65 P9200 Q5020 B-4 (Couple reduction at 20% on B axe enabling with explorative run of 4 mm, the reduction can change from 20 to 50%)

M268(Closing back spindle jaws)

G4 U0.3 (Time of pause for piece exchange)

M69 (Opening jaws spindle)

G94 (Enabling feed in mm/min)

G91 (Enabling movement incremental co-ordinates)

G1 B3 F200 (Incremental displacement of B axe of 3 mm)

G90 (Recall of absolute movement co-ordinates)

G65 P9200 Q 5100 (recall nominal couple)

G0 B0 (Positioning back spindle for further machining)

M72 (Synchronism between spindles not enabled)

G95 (Recall feed in mm/turn)

M11 (Spindle and back spindle rotation with open jaws not enabled)

.... (Machining side back spindle)

6.4 “O9100” - WORKPIECE EXCHANGE WITH PARTING OFF

This is a sub-program that manages the workpiece change-over between spindles machining from bars. For this reason the cutting is performed with a parting off tool. This sub-program is used when, machining the bar, the workpiece is taken up on the back spindle to machine the second part. At the end of the cycle the workpiece with the useful length will remain on the main spindle .

NOTE: ALL VARIABLES ARE TO BE ENTERED INTO THE PROGRAM

Variables to be entered:

X	#24	AXIS X SAFETY DIMENSION
V	#22	AXIS B RAPID APPROACH
B	#2	AXIS B POSITION ON WORKPIECE
E	#8	FEED FOR POSITIONING
W	#23	LENGTH OF FINISHED WORKPIECE
T	#20	PARTING OFF TOOL NUMBER
I	#4	PARTING OFF TOOL WIDTH
K	#6	MACHINING ALLOWANCES ON FACES
D	#7	PARTING OFF START DIAMETER
U	#21	PARTING OFF END DIAMETER
S	#19	VT FOR PARTING OFF
M	#13	DIRECTION OF ROTATION 3/ 4 FOR PARTING OFF > M3/M4
F	#9	PARTING OFF FEED
H	#11	REVS LIMITATION
C	#3	DEPTH OF RADIAL CUT FOR CHIP BREAKAGE
Q	#17	RADIAL SEPARATION FOR CHIP BREAKAGE
R	#18	RECOVERY COLLET CHUCKS CLEARANCE
A	#1	SPINDLE DE-PHASING ANGLE
Z	#26	INCREMENTAL VALUE FOR MECHANICAL STROKE
Y	#25	TORQUE VALUE MIN. 20 MAX. 50

Variables for internal calculations:

#27, #28, #29, #30

Description:

The sub-program is run as follows:

The spindles start to rotate in synchronism (M70) at approx. 50 rpm in the direction defined by variable "M", the parting off tool defined in variable "T" is brought to working position.

The origin used is that which was active before entering the sub-program

The machine opens the back spindle jaws, in rapid traverse it positions X at the dimension specified in the "X" variable, Z at zero and B (back spindle) at the value for approach to the workpiece defined in variable "V" (this value requires verification by manually bringing the back spindle near to the workpiece on the main spindle leaving a space between the two spindles equivalent to the length of a finished workpiece, reading the value of the current position of axis B on the monitor. This value is then inserted in variable "V"). A further reduced feed ("E"= feed in mm/min) of the back spindle is made, to the part holder value defined in variable "B". This variable is defined in two different ways according to whether the back spindle is positioned on a mechanical stop or not.

If the rest on mechanical stop is not used (parameter "Z" at zero) the value is to be found by manually bringing the back spindle on the gripping point, reading the value of axis B current position on the monitor. This value is to be entered in variable "B". If the rest on mechanical stop is used the value found (using the same method as described above – bringing the back spindle manually onto the mechanical stop) must be increased by 1 or 2 mm before being inserted in variable "B" (E.g.: Value read on monitor B-255.5; Value inserted in variable "B"=-254.5). The back spindle makes an exploratory stroke of the value set in parameter "Z" (negative value) within which it should rest on the stop (at the torque set in parameter "Y", min. value 20, max. 50) Otherwise the machine will cut off with an operating error.

Variable "A" sets a de-phasing in degrees between the main spindle and the back spindle (used, for example, to work on hexagonal bars).

The displacement between spindle 1 and spindle 2 refers to function M19 and is obtained by bringing spindle 1 to M19 S0 and spindle 2 to M219 S.. (desired value); value M219 S.. for spindle 2 is to be inserted in variable "A"

The part is gripped by the back spindle, released from the main spindle and extracted by a length that depends on variables "W", "I" and "K".

The main spindle jaws close and the parting off of the workpiece takes place, starting from the diameter defined in variable "D", finishing at the value defined in variable "U" at a feed in mm/rev. defined in variable "F" with Vt in m/min defined in variable "S". For axis Z, the parting off takes place leaving the value of machining allowance for facing "K" on both the main spindle and the back spindle.

Chip breakage can be performed during the parting off, using depth of cut parameters "C" and radial separation "Q". If this possibility is not used, it is sufficient to insert a higher value than the radial parting off depth in "C".

It is also possible to recover any backlash caused by the use of double cone collet chucks, by inserting a value in mm from 0 to 1 in "R", which is recovered before parting off with the bar gripped between the two spindles.

Separation takes place first along axis X at the value defined in variable "X" then axes B and Z simultaneously at the values at which turret rotation took place.

The spindle synchronism is disabled, the main spindle rotation is stopped and a rotation of about 500 rpm is set for the back spindle which remains active when coming out of the sub-program.

At the end of this cycle the workpiece will be removed from the main spindle at the starting value, so that this is ready to start work on a new part.

6.5 “ O9101” - WORKPIECE CHANGE-OVER WITH PARTING OFF, WITHOUT EXTRACTION

This is a sub-program that manages the workpiece change-over between spindles working from a bar, for this reason the cut is made with a parting off tool. This sub-program is used when, working on a bar, the workpiece is taken onto the back spindle to work on the second part. At the end of the cycle, the push bar conveyor or the back spindle is used to extract the workpiece with a useful length for machining

NOTE : ALL THE VARIABLES MUST BE INSERTED INTO THE PROGRAM

Variables to be set:

X	#24	SAFETY DIMENSION AXIS X
V	#22	RAPID TRAVERSE AXIS B
B	#2	AXIS B POSITIONING ON WORKPIECE
E	#8	FEED FOR POSITIONING
W	#23	LENGTH OF FINISHED WORKPIECE
T	#20	PARTING OFF TOOL NUMBER
I	#4	WIDTH OF PARTING OFF TOOL
K	#6	MACHINING ALLOWANCE ON FACES
D	#7	PARTING OFF START DIAMETER
U	#21	PARTING OFF END DIAMETER
S	#19	VT FOR PARTING OFF
M	#13	DIRECTION OF ROTATION 3/4 FOR PARTING OFF > M3/M4
F	#9	PARTING OFF FEED
H	#11	REVS LIMITATION
C	#3	CHIP BREAKAGE RADIAL CUT DEPTH
Q	#17	CHIP BREAKAGE RADIAL SEPARATION
R	#18	COLLET CHUCKS CLEARANCE TAKE-UP
A	#1	SPINDLES DE-PHASING ANGLE
Z	#26	MECHANICAL STOP INCREMENTAL VALUE
Y	#25	TORQUE VALUE MIN. 20 MAX. 50

Variables for internal calculations:

#27, #28, #29, #30, #31, #32

Description:

The sub-program is run as follows:

The spindles start to rotate in synchronism (M70) at approx. 50 rpm in the direction defined by variable "M", the parting off tool defined in variable "T" is brought to working position .

The origin used is that which was active before entering the sub-program.

The machine opens the back spindle jaws, in rapid traverse it positions X at the dimension specified in the "X" variable, Z at zero and B (back spindle) at the value for approach to the workpiece defined in variable "V" (this value requires verification by manually bringing the back spindle near to the workpiece on the main spindle leaving a space between the two spindles equivalent to the length of a finished workpiece, reading the value of the current position of axis B on the monitor. This value is then inserted in variable "V"). A further reduced feed ("E"= feed in mm/min) of the back spindle is made, to the part holder value defined in variable "B". This variable is defined in two different ways according to whether the back spindle is positioned on a mechanical stop or not.

If the rest on mechanical stop is not used (parameter "Z" at zero) the value is to be found by manually bringing the back spindle on the gripping point, reading the value of axis B current position on the monitor. This value is to be entered in variable "B". If the rest on mechanical stop is used the value found (using the same method as described above – bringing the back spindle manually onto the mechanical stop) must be increased by 1 or 2 mm before being inserted in variable "B" (E.g.: Value read on monitor B-255.5; Value inserted in variable "B"=-254.5). The back spindle makes an exploratory stroke of the value set in parameter "Z" (negative value) within which it should rest on the stop (at the torque set in parameter "Y", min. value 20, max. 50) Otherwise the machine will cut off with an operating error.

Variable "A" sets a de-phasing in degrees between the main spindle and the back spindle (used, for example, to work on hexagonal bars).

The displacement between spindle 1 and spindle 2 refers to function M19 and is obtained by bringing spindle 1 to M19 S0 and spindle 2 to M219 S.. (desired value); value M219 S.. for spindle 2 is to be inserted in variable "A"

The part is gripped by the back spindle, released from the main spindle and extracted by a length that depends on variables "W", "I" and "K".

The main spindle jaws close and the parting off of the workpiece takes place, starting from the diameter defined in variable "D", finishing at the value defined in variable "U" at a feed in mm/rev. defined in variable "F" with Vt in m/min defined in variable "S". For axis Z, the parting off takes place leaving the value of machining allowance for facing "K" on both the main spindle and the back spindle.

Chip breakage can be performed during the parting off, using depth of cut parameters "C" and radial separation "Q". If this possibility is not used, it is sufficient to insert a higher value than the radial parting off depth in "C".

It is also possible to recover any backlash caused by the use of double cone collet chucks, by inserting a value in mm from 0 to 1 in "R", which is recovered before parting off with the bar gripped between the two spindles.

Separation takes place first along axis X at the value defined in variable "X" then axes B and Z simultaneously at the values at which turret rotation took place.

The spindle synchronism is disabled, the main spindle rotation is stopped and a rotation of about 500 rpm is set for the back spindle which remains active when coming out of the sub-program.

At the end of this cycle the workpiece will be removed from the main spindle with the minimum required for parting off, to work on a new piece either a new extraction is needed or the use of the push-bar conveyor and the reference pad will bring the workpiece to the correct position.

6.6 “ O9102” - WORKPIECE CHANGE-OVER WITHOUT PARTING OFF

This is a sub-program that manages workpiece changeover between spindles working from a bar section.

Therefore there is no parting off operation.

NOTE : ALL THE VARIABLES MUST BE INSERTED INSIDE THE PROGRAM

Variables to be set:

X	#24	SAFETY DIMENSION AXIS X
V	#22	RAPID TRAVERSE AXIS B
B	#2	AXIS B POSITIONING ON WORKPIECE
E	#8	FEED FOR POSITIONING
M	#13	DIRECTION OF ROTATION 3/4 FOR SYNCHRONISM > M3/M4
A	#1	SPINDLES DE-PHASING ANGLE
Z	#26	MECHANICAL STOP INCREMENTAL VALUE
Y	#25	TORQUE VALUE MIN. 20 MAX. 50

Variables for internal calculations:

#28

Description:

The sub-program is run as follows:

The spindles start to rotate in synchronism (M70) at approx. 50 rpm in the direction defined by variable “M”.

The origin used is that which was active before entering the sub-program.

The machine opens the back spindle jaws, in rapid traverse it positions X at the dimension specified in the “X” variable, Z at zero and B (back spindle) at the value for approach to the workpiece defined in variable “V” (this value requires verification by manually bringing the back spindle near to the workpiece on the main spindle leaving a space between the two spindles equivalent to the length of a finished workpiece, reading the value of the current position of axis B on the monitor. This value is then inserted in variable “V”). A further reduced feed (“E”= feed in mm/min) of the back spindle is made, to the part holder value defined in variable “B”. This variable is defined in two different ways according to whether the back spindle is positioned on a mechanical stop or not.

If the rest on mechanical stop is not used (parameter “Z” at zero) the value is to be found by manually bringing the back spindle on the gripping point, reading the value of axis B current position on the

monitor. This value is to be entered in variable "B". If the rest on mechanical stop is used the value found (using the same method as described above – bringing the back spindle manually onto the mechanical stop) must be increased by 1 or 2 mm before being inserted in variable "B" (E.g.: Value read on monitor B-255.5; Value inserted in variable "B"=-254.5). The back spindle makes an exploratory stroke of the value set in parameter "Z" (negative value) within which it should rest on the stop (at the torque set in parameter "Y", min. value 20, max. 50) Otherwise the machine will cut off with an operating error.

Variable "A" sets a de-phasing in degrees between the main spindle and the back spindle (used, for example, to work on hexagonal bars).

The displacement between spindle 1 and spindle 2 refers to function M19 and is obtained by bringing spindle 1 to M19 S0 and spindle 2 to M219 S.. (desired value); value M219 S.. for spindle 2 is to be inserted in variable "A"

The part is gripped by the back spindle, released by the main spindle.

The back spindle jaws close, after which those of the main spindle open.

Synchronism is disabled and spindle rotation is stopped.

7.0 MACHINING WITH “AXIS C” AND MOTOR DRIVEN TOOLS

Axis C is an option used to program spindle movements intended as angle movements made with programmable feed.

This means that the spindle no longer responds to S functions (rpm.) or M functions (direction of rotation) but becomes an axis to all effects, programmed with address “C” (or “A” in machines fitted with back spindle option).

Therefore, with axis C it is possible to drill holes, cut shapes (keys, eccentricities, undercutting, cams etc..) using certain tools that are referred to as motor driven tools.

7.1 MOTOR DRIVEN TOOLS

The “axis C” option requires the use of special turrets to handle the motor driven modules.

The motor driven modules are axial or radial tool holders upon which the tools for milling, drilling holes and tapping are mounted.

The standard motor driven tools are divided into two groups:

- Axial motor driven modules for front machining
- Radial motor driven modules used to machine on the workpiece diameter

To activate or deactivate the module rotation the following functions are used:

- M303 Clockwise rotation of motor driven module
- M304 Anti-clockwise rotation of motor driven module
- M305 Stop rotation of motor driven module
- S..... Rpm set for motor driven module
- G94 Feed set in mm/min.

The motor driven modules can be mounted in any turret position.

Function S.... corresponds to the actual rpm of the turret motor, therefore it is indispensable to know the module transmission ratio (the modules supplied by Graziano SPA have a 1:1 ratio).

NOTE. It is important that function S.... is written in the block with the direction of rotation of the motor driven module (M303 or M304). This block must not contain other instructions.

Example:

N17

N18 M304 S2000 ; Motor driven module rpm and direction of rotation

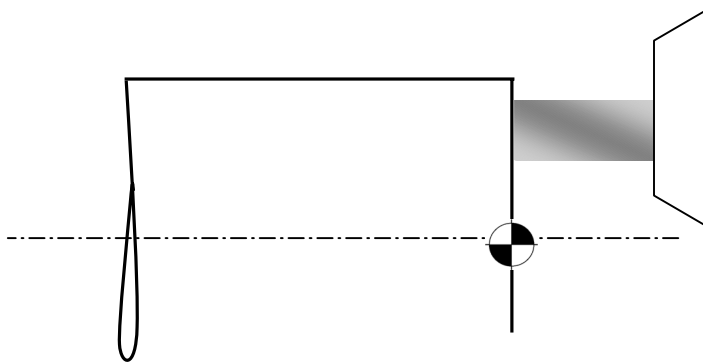
Example of functions used for motor driven modules:

N17
N18 T101 ; Call up for turning tool
N19
N20 ; Turning
N21
N22 T202 ; Call up for milling tool
N23 G54 ; Origin activation
N24 M303 S1000 ; Module rpm and direction of rotation
N25 G94 F500 ; Feed mm/min set.
N26
N27 ; Machining with motor driven module
N28
N29 M305 ; Stop module rotation
N30 T303 ; Call up for turning tool
N31 G95 ; Feed mm/rev. set
N32

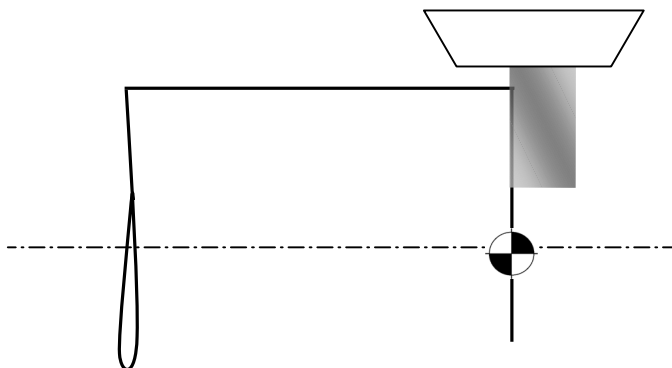
7.2 MOTOR DRIVEN TOOLS RESET

All the tools mounted on motor driven modules (cutters, bits, tapping bits etc.) reset with the same procedure used for normal turning tools.

Axial motor driven modules => They reset only along axis Z, the tool length along axis X must be zero (X0) because these tools are co-axial with the turret "zero" position. The reset procedure is described in the Concise Guide for Operator, chapter 14 – TOOL RESET.



Radial motor driven modules => These reset on both axes (X and Z) like a standard lathe tool. When resetting on axis Z it must be decided whether to reset the tool in relation to the milling machine rotation axis or on the side of the actual milling machine. The reset procedure is described in the Concise Guide for Operator, chapter 14 – TOOL RESET



7.3 AXIS C

The axis C option is activated by functions M37 and G28 C0 (M237 and G28 A0 in the case of back spindle option) whereas to leave this option and return to turning mode it is sufficient to program function M36 (M236 for back spindle option).

Example:

```
N26 .....  
N27 M37           ; Enable axis C on main spindle  
N28 G28 C0        ; Axis C reference  
N29 T202          ; Call up tool  
N30 G54           ; Activation of work origin  
N31 M303 S1000    ; Rpm and direction of rotation activation  
N32 G0 X... Z... C0 ; Axis C positioning  
N33 G94 F500      ; Feed mm/min set.  
N34 .....        ; Work with motor driven module  
N35 .....  
N36 M305          ; Stop rotation of rotating module  
N37 M36           ; Disable axis C on main spindle  
N38 G95           ; Feed mm/rev set.  
N39 .....
```

The block containing function G28 C0 (or G28 A0) must not contain other instructions.

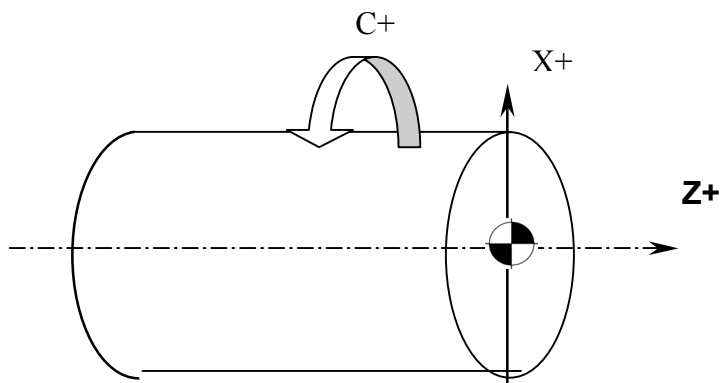
The axis C option can be used in three different ways:

- Real co-ordinates.
- Imaginary co-ordinates (G112).
- Cylindrical interpolation (G107).

7.4 PROGRAMMING IN REAL CO-ORDINATES

When functions M37 and G28 C0 (M237 and G28 A0 on machines with back spindle option) the machine prepares to work in “real co-ordinates”.

X..... Z..... C (A).....



Where :

- X => Absolute co-ordinate of axis X, is to be programmed with a diametrical value.
- Z => Absolute co-ordinate of axis Z.
- C => Co-ordinate for axis C positioning on main spindle.
- A => Co-ordinate for axis C positioning on back spindle.

The positive direction corresponds to the spindle direction of rotation (M4). Code C is programmed as an angle value in degrees up to a maximum of the third decimal digit.

Example: N51 G0 C180.123

Axis C, used in real co-ordinates makes it possible to drill front and radial holes, make front and radial tapping, key seats, front concentric slots and helical milling on the workpiece outer diameter.

To make an incremental displacement of axis C, function H... can be used.

Example: N32 G0 H90 (axis C moves incrementally by 90 degrees in relation to the point where it is currently positioned)

Code H is also used to make axis C movements with a value over 360° (spirals, threads or to use the motor driven module for grinding combined with the spindle rotation)

Example :N32 G1 H3600 (axis C moves incrementing by 3600 degrees, i.e. making 10 spindle turns)

7.5 USE OF SPINDLE BRAKE

The machines with the axis C option have a brake which acts on a disk integral with the spindle, preventing rotation due to any machining stress. The functions to manage the brake are:

- M20 ⇒ Activation of main spindle brake
- M21 ⇒ Deactivation of main spindle brake

In machines with the back spindle option these instructions are also used:

- M220 ⇒ Activation of back spindle brake
- M221 ⇒ Deactivation of back spindle brake

The use of the brake is advised for milling and drilling holes with spindle stationary, that is, when axis C is used as spindle orientation (divider type) to ensure better system stability (for example working on holes, tapping, key seats etc.) . It is not possible to program the spindle rotation with the brake on (M20 active) or when programming in imaginary co-ordinates (G112 or G107) since the axis interpolation requires spindle movement.

7.6 "G83" FRONT DRILLING CYCLE

Function "G83" activates the front drilling cycle with motor driven tools.

With this function the bit makes a series of cuts, of the required size, undercutting or breaking the chips and returning at the end of the cycle, in rapid transverse, to the starting point or to point R.

The front drilling cycle can contain these codes:

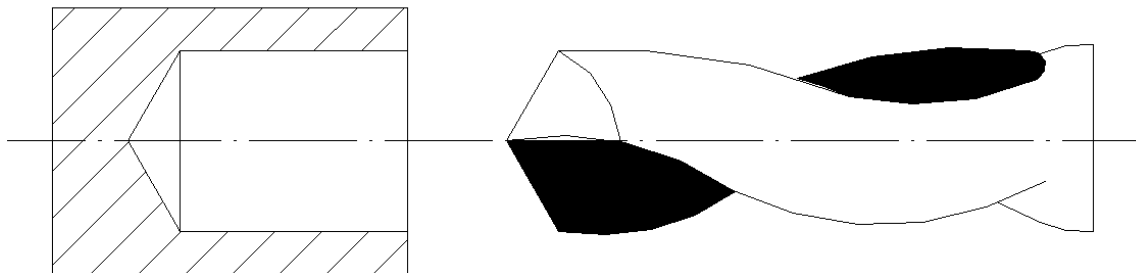
- Z => Absolute value of end of drilling
- F => Drilling feed (expressed in mm/minute)
- Q => Cut depth (in thousandths)
- P => Pause at bottom of hole (in thousandths of seconds)
- R => Incremental distance from starting point of cycle to starting point of hole.

Out of all the parameters described above, the only ones which are compulsory are Z (end of drilling value) and F (drilling speed), all the other parameters are only to be programmed if actually used.

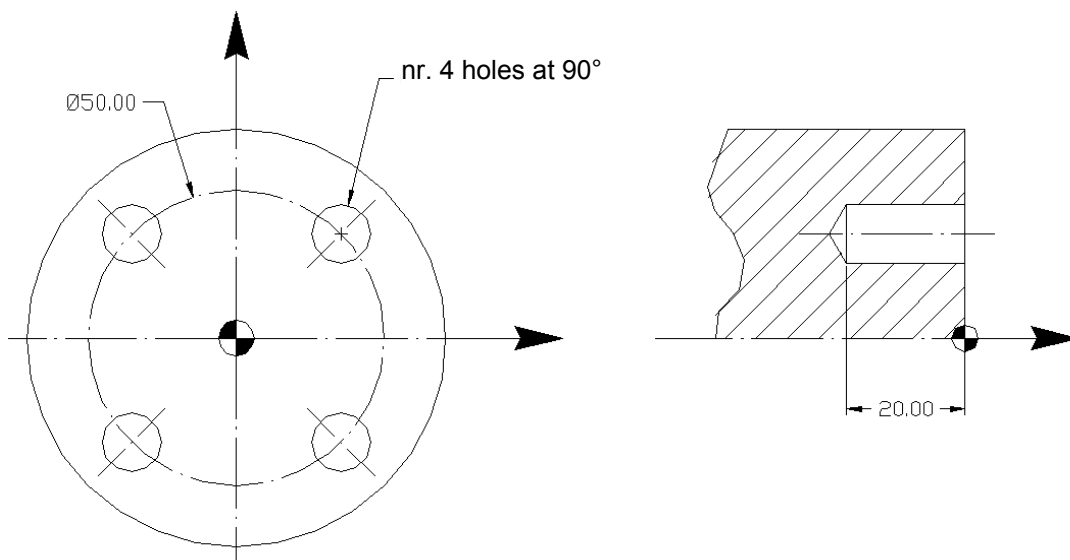
If R parameter is used, the distance between the starting point of cycle and the starting point of hole is executed in rapid. Eventual discharges (parameter Q) occur at the starting point of hole, while at the end of drilling the tip comes back to the starting point of cycle.

If P parameter is used the pause is executed only on final point of drilling.

To leave the drilling cycle function G80 must be programmed, or any G function of the 01 group, i.e. G0, G1, G2, or G3.



Example: drilling of 4 axial holes, depth 20 mm. diameter 50



N34TURNING

N35 M37

N36 G28 C0

N37 T101 (AXIAL BIT)

N38 G54

N39 M303 S2000

N40 G94

N41 G0 X50 Z5 M8

N42 C0 M20

N43 G83 Z-20 F100

N44 C90 M20

N45 C180 M20

N46 C270 M20

N47 G80

N48 G0 X200 Z200 M21

N49 M305

N50 M36

N51 G95

N52 M30

NOTE. FUNCTIONS M20/M21 FOR THE USE OF THE SPINDLE BRAKE ARE OPTIONAL.

Codes Q, P and R , if not used, need not be written.

This cycle can be used with chip breakage or undercutting, depending on the value of parameter 5101 bit 2 (if it is 0 chip breakage, if it is 1 chip undercutting) by default this bit is set to 1 for chip undercutting.

Parameter 5114 determines:

- in the case of chip undercutting, the distance at which the bit must stop in relation to the last point reached when re-entering the hole after undercutting.
- in the case of chip breaking, how much the bit must back off between one cut and the next for drilling

7.7 "G87" RADIAL DRILLING CYCLE

Function "G87" activates the side radial cycle with motor driven tools.

With this function the bit makes a series of cuts, of the required size, undercutting or breaking the chip and returning with a rapid traverse at the end of cycle to the starting point or to point R.

The radial drilling cycle can contain these codes:

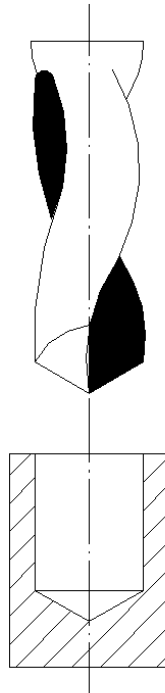
- X => Absolute value at end of drilling
- F => Drilling feed (in mm/minute)
- Q => Depth of cut (in thousandths)
- P => Pause at bottom of hole (in thousandths of seconds)
- R => Incremental distance from starting point of cycle to starting point of hole

Out of all the parameters described above, the only ones which are compulsory are X (end of drilling value) and F (drilling feed), all the other parameters are only to be programmed if actually used.

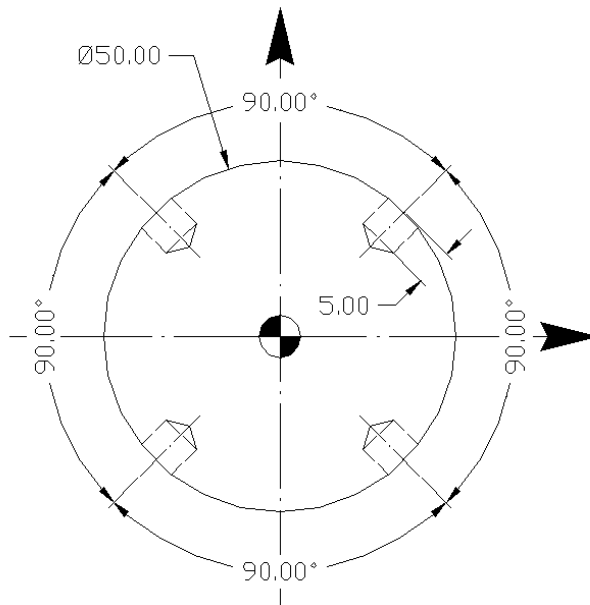
If R parameter is used the distance between the starting point of cycle and the starting point of hole is executed in rapid. Eventual discharges (parameter Q) occur at the starting point of hole, while at the end of drilling the point comes back to the starting point of cycle.

If P parameter is used the pause is executed only on final point of drilling.

To leave the drilling cycle function G80 must be programmed, or any G function of the 01 group, i.e. G0, G1, G2, or G3.



Example: 4 radial holes at 20 mm from the workpiece zero



N34TURNING

N35 M37

N36 G28 C0

N37 T101 (RADIAL BIT)

N38 G54

N39 M303 S2000

N40 G94

N41 G0 X55 Z5

N42 Z-20 M8

N43 C0 M20

N44 G87 X40 F100

N45 C90 M20

N46 C180 M20

N47 C270 M20

N48 G80

N49 G0 X200 Z200 M21

N50 M305

N51 M36

N52 G95

N53 M30

NOTE: FUNCTIONS M20/M21 TO USE THE SPINDLE BRAKE ARE OPTIONAL.

If not used, codes Q, P and R need not be written.

This cycle can be used with chip breakage or undercutting, depending on the value of parameter 5101 bit 2 (if it is 0 chip breakage, if it is 1 chip undercutting) by default this bit is set to 1 for chip undercutting.

Parameter 5114 determines:

- in the case of chip undercutting, the distance at which the bit must stop in relation to the last point reached when re-entering the hole after undercutting.
- in the case of chip breaking, how much the bit must back off between one cut and the next for drilling

7.8 “O9103” FRONT TAPPING SUB-PROGRAM

Sub-program “9103” activates the axial tapping cycle.

With this function the tapping tool enters with a feed equal to the tapping pitch, reverses the module rotation, followed by simultaneous acceleration of the motor driven tool and the axis then the return to starting point

The axial tapping cycle contains these codes:

- Z => End of tapping absolute value
- F => Tapping pitch (in mm/rev.)
- S => Motor driven tool rpm
- M => Module direction of rotation when entering (303 or 304)

On machines fitted with tool holder disk with axial seats (GT400M, GT500M and GT700M), to use the tapping sub-program another two functions must be enabled:

- M341 Engagement of module to the turret disk PTO units
- M340 Disengagement of the module to the turret disk PTO units

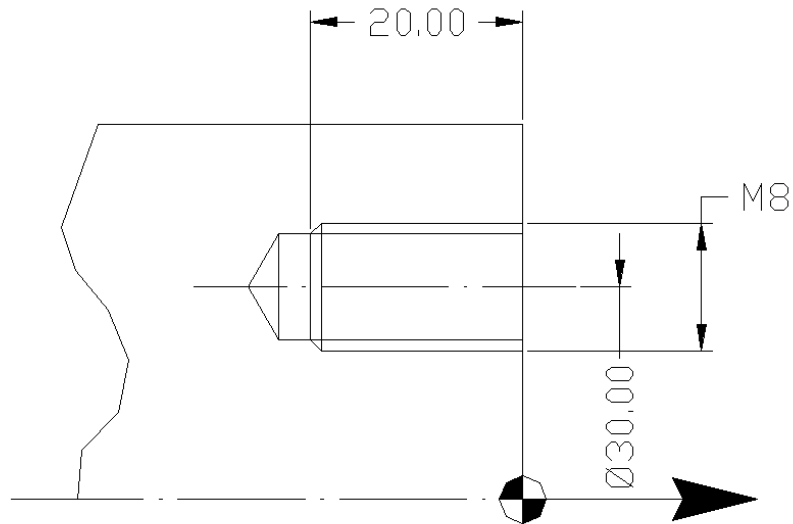
Delivery of coolant only takes place through function M7.

Disabling of coolant delivery is through function M9.

Sub-program “9103” can be called up in single mode (by function G65) or in modal mode (by function G66 cancelled at end of cycle by G67).

NOTE. GRAZIANO SPA ADVISES USE OF COMPENSATED COLLETS IN TAPPING WITH DRIVEN TOOLS.

Example of single call-up (tapping of one hole only):



N15 TURNING

N16 M37

N17 G28 C0

N18 T606 (AXIAL TAPPING M8)

N19 G54

N20 M341 (only for axial disks)

N21 G94

N22 G0 X30 Z5 M7

N23 C0 M20

N24 G65 P9103 Z-20 M303 F1.25 S200

N25 M340 (only for axial disks)

N26 G0 X150 Z50 M21

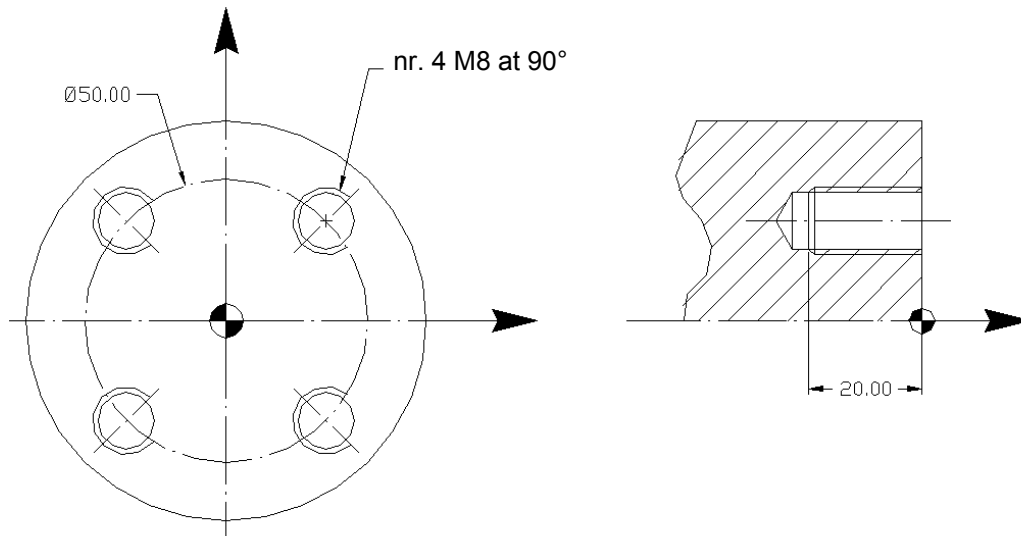
N27 M36

N28 G95 M9

N29 M30

NOTE: FUNCTIONS M20/M21 FOR USE OF THE SPINDLE BRAKE ARE OPTIONAL..

Example of modal call-up (tapping of several holes):



N15 TURNING

N16 M37

N17 G28 C0

N18 T606 (AXIAL TAPPING M8)

N19 G54

N20 M341 (only for axial disks)

N21 G94

N22 G0 X50 Z5 M7

N23 G66 P9103 Z-20 M303 F1.25 S200

N24 C0 M20

N25 C90 M20

N26 C180 M20

N27 C270 M20

N28 G67

N29 M340 (only for axial disks)

N30 G0 X150 Z50 M21

N31 M36

N32 G95 M9

N33 M30

NOTE: FUNCTIONS M20/M21 TO USE THE SPINDLE BRAKE ARE OPTIONAL.

7.9 “O9104” RADIAL TAPPING SUB-PROGRAM

Sub-program “9104” activates the radial tapping cycle.

With this function the tapping tool enters with a feed equal to the tapping pitch, reverses the module rotation, followed by simultaneous acceleration of the motor driven tool and the axis then the returns to starting point.

The axial tapping cycle contains these codes:

- X => End of tapping absolute value
- F => Tapping pitch (in mm/rev.)
- S => Motor driven tool rpm
- M => Module direction of rotation when entering (303 or 304)

On machines fitted with tool holder disk with axial seats (GT400M, GT500M and GT700M) to use the tapping sub-program another two functions must be enabled:

- M341 Engagement of module to the turret disk PTO units
- M340 Disengagement of the module to the turret disk PTO units

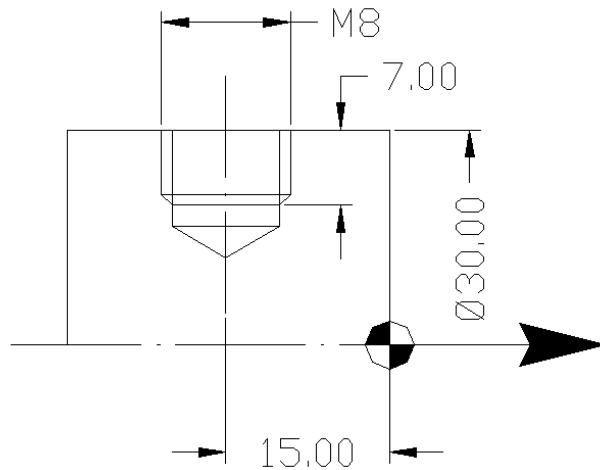
Delivery of coolant only takes place through function M7.

Disabling of coolant delivery is through function M9.

Sub-program “9104” can be called up in single mode (by function G65) or in modal mode (by function G66 eliminated at the end of cycle by G67).

NOTE. GRAZIANO SPA ADVISES USE OF RIGID COLLETS IN TAPPING WITH DRIVEN TOOLS.

Example of single call-up (tapping of one hole only):



N15 TURNING

N16 M37

N17 G28 C0

N18 T707 (RADIAL TAPPING M8)

N19 G54

N20 M341 (only for axial disks)

N21 G94

N22 G0 X35 Z-15 M7

N23 C0 M20

N24 G65 P9104 X16 M303 F1.25 S200

N25 M340 (only for axial disks)

N26 G0 X150 Z50 M21

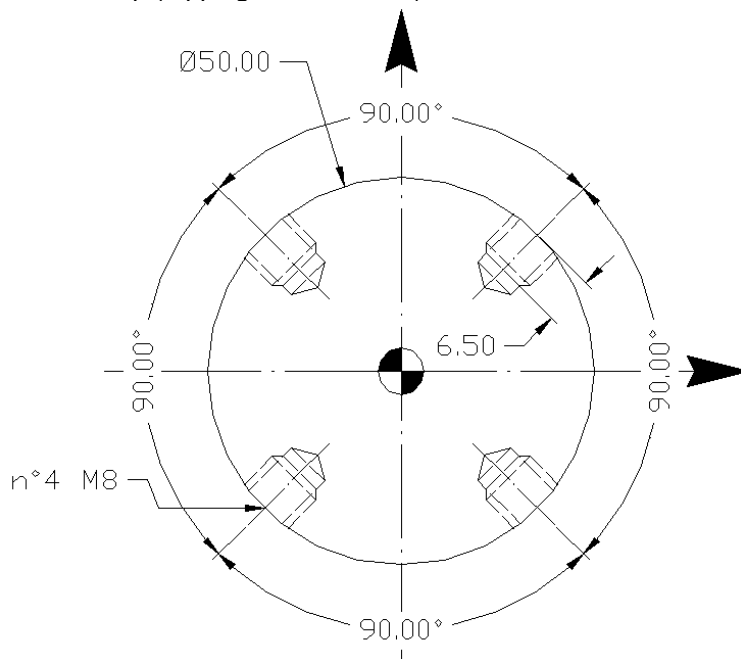
N27 M36

N28 G95 M9

N29 M30

NOTE: FUNCTIONS M20/M21 TO USE THE SPINDLE BRAKE ARE OPTIONAL.

Example of modal call-up (tapping several holes):



N15 TURNING

N16 M37

N17 G28 C0

N18 T707 (RADIAL TAPPING M8)

N19 G54

N20 M341 (only for axial disks)

N21 G94

N22 G0 X55 Z-15 M7

N23 G66 P9104 X37 M303 F1.25 S200

N24 C0 M20

N25 C90 M20

N26 C180 M20

N27 C270 M20

N28 G67

N29 M340 (only for axial disks)

N30 G0 X150 Z50 M21

N31 M36

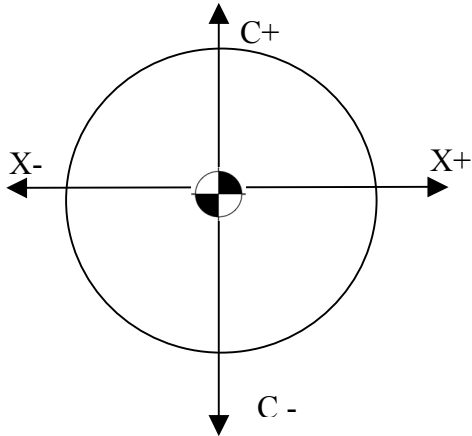
N32 G95 M9

N33 M30

NOTE: FUNCTIONS M20/M21 TO USE THE SPINDLE BRAKE ARE OPTIONAL.

7.10 G112 PROGRAMMING IN IMAGINARY CO-ORDINATES

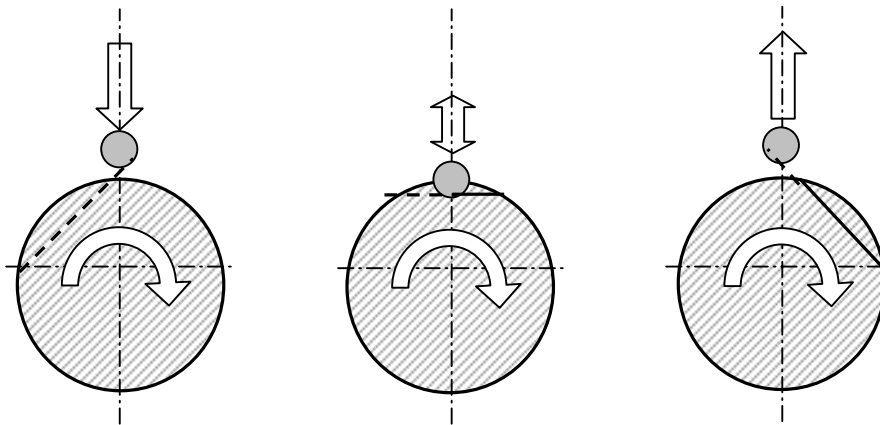
Function G112, used to program on the front surface, transforms the real co-ordinates into imaginary co-ordinates.



The imaginary axes are obtained by interpolating real axes X and C. Therefore with G112 active, the control calculates the feed and the points needed to move the real axes along the imaginary components X C.

It therefore results that every movement of imaginary X and C moves the two real axes.

Example of work trend in imaginary co-ordinates:



Function G112 is programmed in a block without any other instructions.

In imaginary co-ordinates G112 the co-ordinates of C are radial whereas the co-ordinates of X are diametrical.

NOTE: After function G112 has been activated no further rapid traverses are allowed (G0), the origin cannot be moved (from table G54-G59 and program G52) and no corrector change is allowed

The activation of function G112 does not involve movement of the machine axes, and the monitor shows the addresses of the new co-ordinates.

The activation and deactivation functions of the milling radius offset (G41, G42 e G40) are only allowed after function G112 has been activated.

When the milling operation has been terminated, before the separation and release of axis C, it is necessary to return to the real co-ordinates by activating function G113.

Example of passage from turning operation to working in imaginary co-ordinates(G112):

```
N14 ....
N15 ....(TURNING OPERATIONS)
N16 ....
N17 M37 (OR M237 FOR BACK SPINDLE)
N18 G28 C0 (OR G28 A0 FOR BACK SPINDLE)
N19 T101
N20 G54
N21 M303 S1000
N22 G94 F500
N23 G0 X100 Z10 C0 (OR Z-10 A0 FOR BACK SPINDLE)
N24 G112 (ENABLE IMAGINARY CO-ORDINATES)
N25 ....
N26 ....
N27 .... (MILLING OPERATIONS)
N28 ....
N29 G113 (RETURN TO REAL CO-ORDINATES)
N30 G0 Z100
N31 M305
N32 M36 (OR M236 FOR BACK SPINDLE)
N33 G95
N34 ....
N35 .... (TURNING OPERATIONS)
N36 ....
```

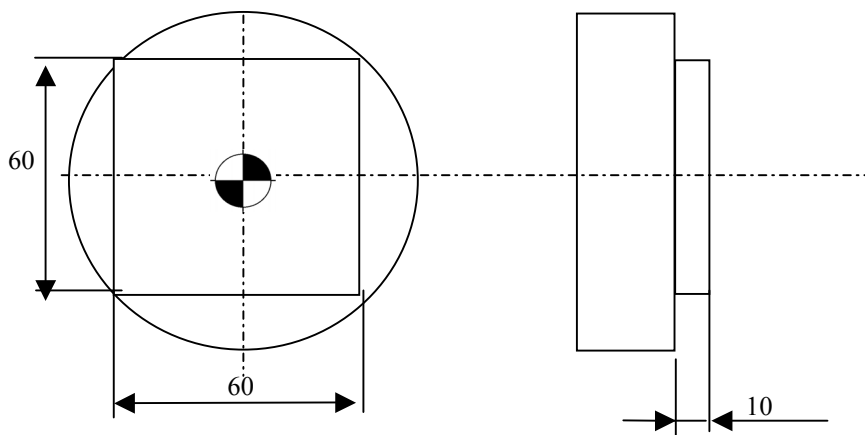
All work in G112 mode is to be carried out with axial motor driven tools.

The cutter/bit must be reset only along axis Z, however, it is necessary to write 0 (zero) in the tool table, in the geometry offset column next to the corrector used.

To obtain a correct result, the cutters/bits must be aligned and centred to the motor driven tool.

Inside the G112 interpolation no fixed drilling or tapping cycles can be used.

Example: Milling operation without using radius offset in G112:



N15 (TURNING OPERATION)

N16

N17 M37

N18 G28 C0

N19 T101

N20 G54

N21 M303 S1500

N22 G94 F1000

N23 G0 X100 Z2 C0 M8

N24 G112

N25 G1 Z-10 F1000

N26 X70 C30 F120

N27 X-60

N28 C-30

N29 X60

N30 C35

N31 Z2 F1000

N32 G113

N33 G0 Z100

N34 M305

N35 M36

N36 G95

N37 M30

7.11 CIRCULAR INTERPOLATION IN G112

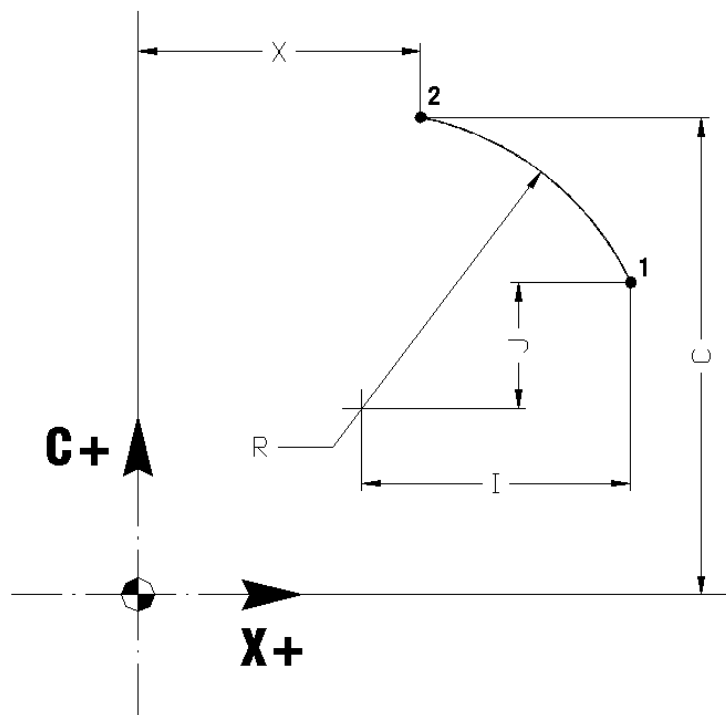
The circular interpolations G2/G3 on the front surface (G112 active) can be programmed in two ways :

- Coupling the value of radius R to the co-ordinates of end of interpolation X and C (method most commonly used).
- Coupling the incremental co-ordinates of the distance from the circle centre to the interpolation starting point I and J to the end of interpolation co-ordinates X and C (I is referred to axis X, J is referred to axis C)

Example:

G2 o G3 X..... C..... R.....

G2 o G3 X..... C..... I..... J.....



Where:

- G2 / G3 => Circular interpolation direction clockwise/anti-clockwise
- X => Co-ordinate of final point along axis X
- C => Co-ordinate of final point along axis C
- R => Radius of circular interpolation
- I => Incremental co-ordinate along axis X
- J => Incremental co-ordinate along axis C

7.12 G41 G42 G40 MILLING RADIUS OFFSET IN G12

Also in milling, as for turning, the tool radius offset can be used .

To do so, it is necessary to enter in the tool table the cutter radius (R) and the tool orientation (T), The value of this orientation can be either T0 or T9 (for the procedure to enter this data see the Concise Guide for Operator).

It is also necessary to insert in the program functions G41 or G42 to activate the offset and G40 for the deactivation.

Functions G41 and G42 are used to define the position of the cutter as to the workpiece:

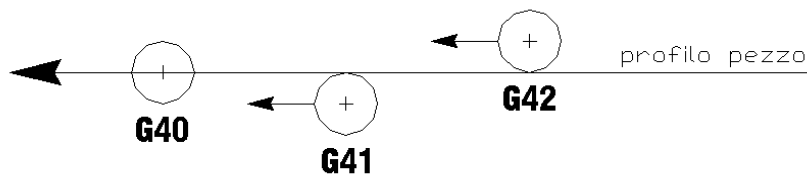
G41 => Workpiece on RIGHT of cutter

G42 => Workpiece on LEFT of cutter

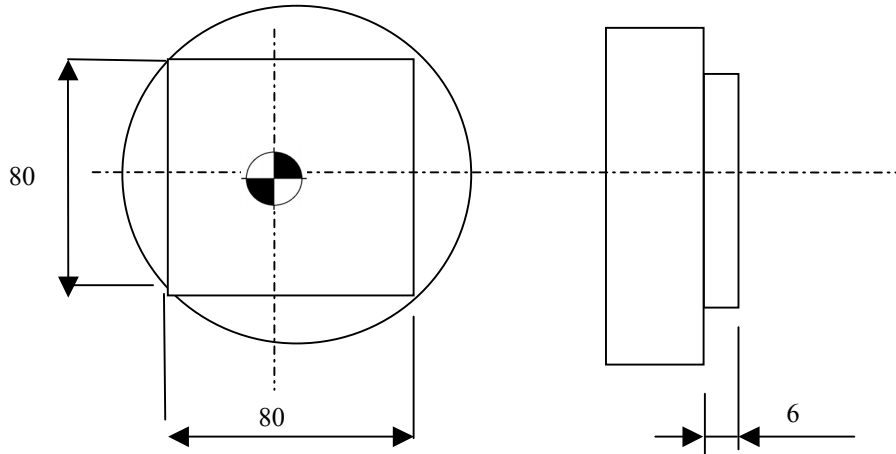
Function G40 DEACTIVATES the milling radius offset, with this function active, the described profile is travelled from the cutter centre.

NOTE: It is recommended to activate (G41 or G42) and deactivate (G40) the milling radius offset at a distance greater than the value of the radius of the cutter used.

It is best to start and interrupt the work with milling radius offset not at the exact point of the beginning of the work, but on an extension of the profile.



Example of milling operation with radius offset in G112:



N16 (TURNING OPERATION)

N17 M37

N18 G28 C0

N19 T101

N20 G54

N21 M303 S1500

N22 G94 F1000

N23 G0 X100 Z2 C0 M8

N24 G112

N25 G1 Z-6

N26 X100 C50 F120

N27 G1 G42 X90 C40 (ACTIVATE MILLING RADIUS OFFSET)

N28 X-80

N29 C-40

N30 X80

N31 C45

N32 G40 (DEACTIVATE MILLING RADIUS OFFSET)

N33 Z2 F1000

N34 G113

N35 G0 X200 Z100

N36 M305

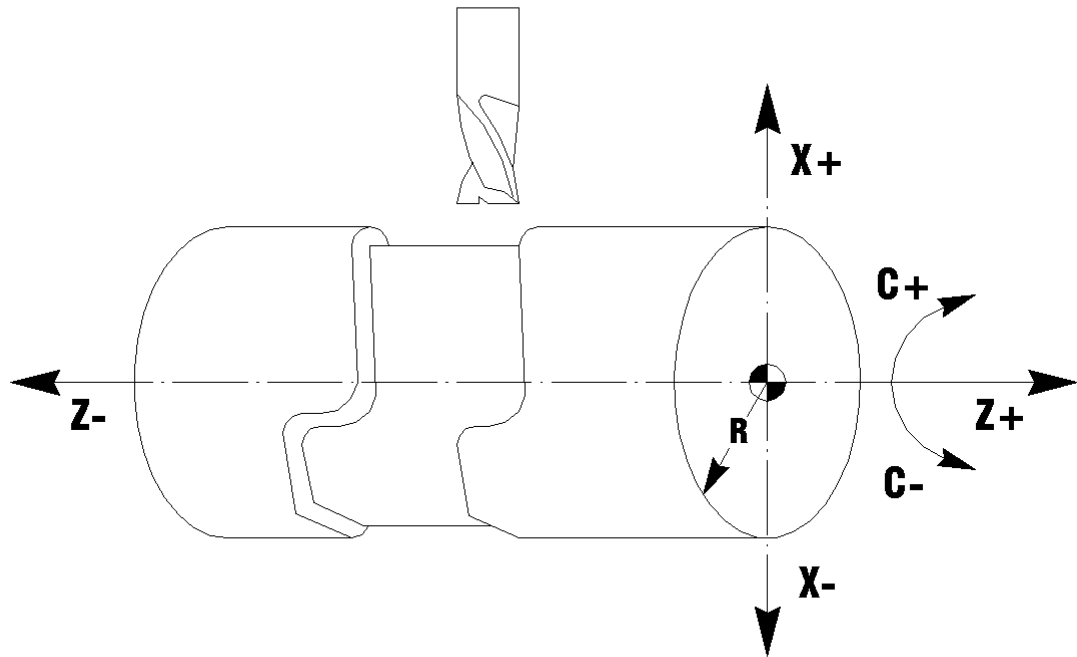
N37 M36

N38 G95

N39 M30

7.13 "G107" CYLINDRICAL INTERPOLATION

The cylindrical interpolation function G107 allows programming taking into consideration the total length of the plane of the side surface of a cylinder; therefore, this is very useful to program splines of cylindrical cams performed on the skirt of the workpiece (interpolating axes Z and C) and using a radial motor driven module.



To enable and disable function G107 the procedure is as follows:

- G1 G18 W0 H0** Specifies that work starts interpolating axis Z with axis C (W and H are the incremental values of Z and C)
- G107 C....** **G107** activates the cylindrical interpolation mode, **C..** specifies the radius of the piece to be worked, it serves for the feed speed calculation G94 F in mm/min according to the milling radius (as the working radius increases the spindle will turn more slowly) The value of C is used also for the calculation of the new transferred profile of the milling radius when the milling radius offset G41 or G42 is activated,

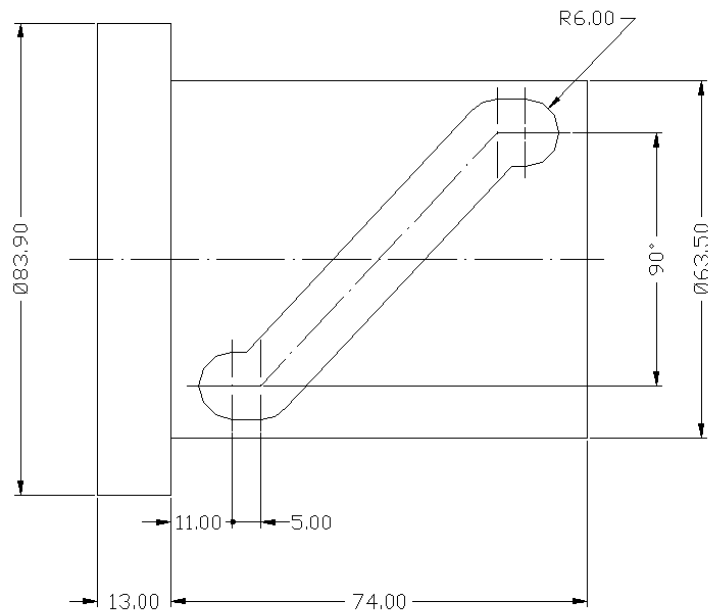
.....

G107 C0 Cancels the cylindrical interpolation G107

The working plane is transformed in this way:

- Functions G107C... and G107 C0 must be written in a block on their own
- After instruction G107C... only functions G1 G2 G3 can be used, direct programming functions ,A ,C etc .cannot be used
- Tool radius offset G41,G42 and G40 must be activated and deactivated inside function G107
- All work in G107 mode is to be carried out with radial motor driven tools.
- For correct machining the cutters/bits are to be aligned and centred as to the motor driven tool.
- Within the G107 interpolation no fixed drilling or tapping cycles can be used.
- Within G107 interpolation no displacement of origin G52 and G54 –G59 is allowed

Example of how to use function G107 (working a piece with diameter 55)



N16 (TURNING OPERATION)

N17 M37 (M237 for back spindle)

N18 G28 C0 (G28 A0 for back spindle)

N19 T101

N20 G54 (G55 for back spindle)

N21 M303 S1500

N22 G94 F1000

N23 G1 G18 W0 H0 (G91 G18 Z0 A0 / G90 for back spindle)

N24 G0 X 70 Z10 C0 M8 (A0 for back spindle)

N25 G107 C27.5

N26 G1 Z-11 F1000

N27 X55 F120

N28 Z- 16

N29 Z-58 C90 (A90 for back spindle)

N30 X70 F1000

N31 X70 F1000

N32 Z2

N33 G 107 C0

N34 G18

N35 G0 X200 Z100

N36 M305

N37 M36 (M236 for back spindle)

N38 M95

N39 M30

7.14 PROGRAMMING WITH REAL Y AXIS

Machines that have the axis Y option can make transverse movements of the turret of 64 mm (from -32 to +32).

Through this option it is therefore possible to make radial machining that is not perpendicular to the centre of the workpiece (out of axis), for example drilling and tapping out of axis in relation to the centre of the workpiece.

Planes milling can be carried out using a radial module (which is physically impossible using the G112 imaginary co-ordinates) providing it is compatible with the ± 32 mm stroke of the turret.

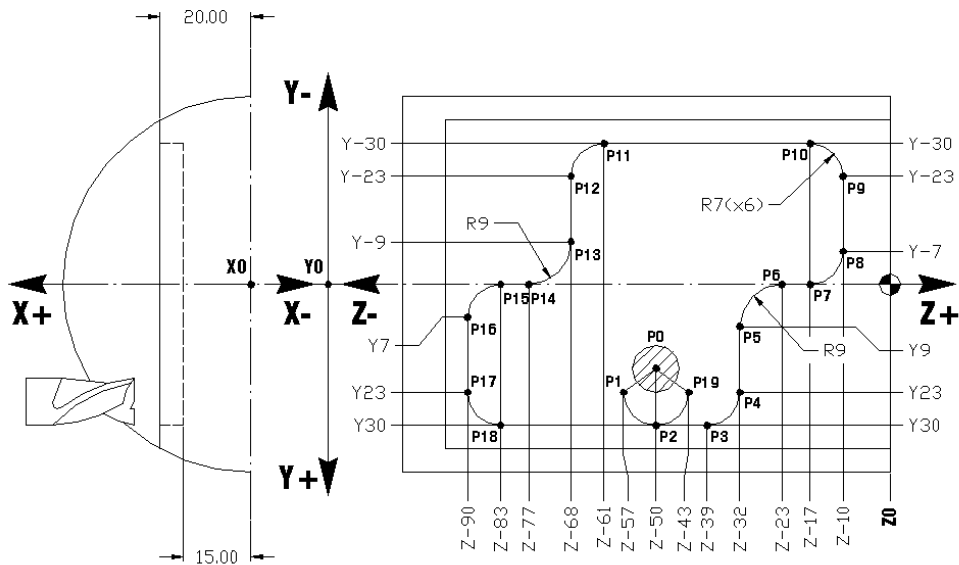
For programming axis Y is treated like another real axis (X,Z,C etc) and will have a positive direction (toward the operator) or a negative direction (toward the machine interior)

The lathe with axis Y is always a machine with a motor driven turret and axis C, therefore the functions already described are valid.

In machines having this option, before rotating the turret it is necessary to reposition axis Y to zero by programming the instruction G0 Y0

If it is required to make a circular interpolation (G2/G3) of axes Y and Z, the work plane where the arc is found (G19) must be specified and after the operation has terminated, return to the normal work plane (G18).

Example of how to use axis Y



N15TURNING

N16 M5

N17 G0 Y0

N18 M37

N19 G28 C0

N20 T101

N21 G94

N22 M303 S1200

N23 G0 X100 Z-50

N24 Y18

N25 X42

N26 G19

N27 G1 X30 F80 (P0)

N28 G1 G41 Z-57 Y23 (P1)

N29 G3 Z-50 Y30 R7 (P2)

N30 G1 Z-39 F150 (P3)

N31 G3 Z-32 Y23 R7 (P4)

N32 G1 Y9 (P5)

N33 G2 Z-23 Y0 R9 (P6)

N34 G1 Z-17 (P7)
N35 G3 Z-10 Y-7 R7 (P8)
N36 G1 Y-23 (P9)
N37 G3 Z-17 Y-30 R7 (P10)
N38 G1 Z-61 (P11)
N39 G3 Z-68 Y-23 R7 (P12)
N40 G1 Y-9 (P13)
N41 G2 Z-77 Y0 R9 (P14)
N42 G1 Z-83 (P15)
N43 G3 Z-90 Y7 R7 (P16)
N44 G1 Y23 (P17)
N45 G3 Z-83 Y30 R7 (P18)
N46 G1 Z-50 (P2)
N47 G3 Z-43 Y23 R7 (P19)
N48 G1 G40 Z-50 Y18 (P0)
N49 G0 X80
N50 G18
N51 G0 Y0 Z100
N52 M305
N53 M36
N54 G95
N55 M30

8.0 BAR MACHINING

We have included in this chapter some examples of programs that use loaders, push bar conveyors, and an example using a pull-bar conveyor in a cycle for machine with and without back spindle.

8.1 EXAMPLE OF MACHINE TOOL LOADER USE WITH BACK SPINDLE

The program example below regards a bar loader connected to a machine with a back spindle, and it is valid for LNS loaders with magazine of the type QUICK LOAD, SPRINT and IEMCA

GOTO 200 (UNCONDITIONED SKIP TO BLOCK N200)

N10 T101 (TOOL FOR BAR REFERENCE)

G54

G97 M3 S50

M10 (ENABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X0 Z5

Z-47

M69 (OPEN SELF CENTRING CHUCK / COLLET CHUCK)

G4 U1 (PAUSE TIME FOR SELF-CENTRING CHUCK/COLLET CHUCK OPENING)

G1 Z0.2 F10

M68 (CLOSE SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

M11 (DISABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X200 Z100

T202 (TOOL FOR MACHINING SPINDLE 1 SIDE)

G54

MACHINING COMPLETE SPINDLE 1 SIDE

G0 B0 (REPOSITIONING BACK SPINDLE FOR WORKPIECE CHANGE-OVER)

G65 P9100 (WORKPIECE CHANGE-OVER WITH PARTING OFF MACRO)

G0 B0 (REPOSITIONING BACK SPINDLE FOR WORKPIECE CHANGE-OVER)

T222 (TOOL FOR MACHINING SPINDLE 2 SIDE)

G55

MACHINING COMPLETE SPINDLE 2 SIDE AND WORKPIECE UNLOADING

M62 (PIECE COUNTER INCREMENT)

M1 (OPTIONAL STOP)

N200 IF[104EQ0]GOTO10 (FINISHED BAR CONTINUE, NOT FINISHED SKIP TO N10)

T101 (TOOL FOR NEW BAR REFERENCE)

G54

G97 M3 S50

M10 (ENABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X0 Z5 M9

Z-47

M69 (OPEN SELF-CENTRING CHUCK/COLLET CHUCK)

M67 (STAND-BY FOR LOADING NEW BAR SIGNAL)

M68 (CLOSE SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

M11 (DISABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X200 Z100

T505 (TOOL FOR NEW BAR FACING/PARTING OFF)

G54

G92 S2500

G96 S120 M4

G0 X42 Z0.2 M8

G1 X-1 F0.1

G0 X200 Z200 M9

G0T010 (UNCONDITIONED SKIP TO BLOCK N10)

M30

8.2 EXAMPLE OF MACHINE TOOL LOADER USE WITHOUT BACK SPINDLE

The program example below regards a bar loader connected to a machine without a back spindle, and it is valid for LNS loaders with magazine of the type QUICK LOAD, SPRINT and IEMCA

GOTO 200 (UNCONDITIONED SKIP TO BLOCK N200)

N10 T101 (TOOL FOR BAR REFERENCE)

G54

G97 M3 S50

M10 (ENABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X0 Z5

Z-47

M69 (OPEN SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1 (STAND-BY FOR SELF-CENTRING CHUCK/COLLET CHUCK OPENING)

G1 Z0.2 F10

M68 (CLOSE SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

M11 (DISABLE SPINDLE ROTATION WITH JAWS OPEN)

T202 (COMPLETE MACHINING OF ITEM)

G54

G92 S2500

G96 S120 M4

G0 X42 Z.2 M8

G0 X200 Z200 M9

M62 (PIECE COUNTER INCREASE)

M1 (OPTIONAL STOP)

N200 IF[104EQ0]GOTO10 (FINISHED BAR CONTINUE, NON FINISHED SKIP TO N10)

T101 (TOOL FOR NEW BAR REFERENCE)

G54

G97 M3 S50

M10 (ENABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X0 Z5 M9

Z-47

M69 (OPEN SELF-CENTRING CHUCK/COLLET CHUCK)

M67 (STAND-BY FOR NEW BAR LOADING SIGNAL)

M68 (CLOSE SELF CENTRING CHUCK/COLLET CHUCK)

G4 U1
M11 (DISABLE SPINDLE ROTATION WITH JAWS OPEN)
G0 X200 Z200
T505 (TOOL FOR FACING / PARTING OFF NEW BAR)
G54
G92 S2500
G96 S120 M4
G0 X42 Z0.2 M8
G1 X-1 F0.1
G0 X200 Z200 M9
G0T010 (UNCONDITIONED SKIP TO BLOCK N10)
M30

8.3 EXAMPLE OF MACHINE TOOL BAR-FEEDER CONVEYOR USE WITH BACK SPINDLE

The example below shows the use of a single pipe push-bar conveyor for machine with back spindle.

T101 (TOOL FOR FACING / PARTING OFF NEW BAR)

G54

G97 M3 S50

M10 (ENABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X0 Z5 M9

Z-47

M69 (OPEN SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

G1 Z1 F10

M68 (CLOSE SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

M11 (DISABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X200 Z100

N100 G0 B0

T202 (MACHINING OF PART SPINDLE 1 SIDE)

G54

G92 S2500

G96 S180 M4

G0 X42 Z0 M8

...

G0 X200 Z100

G65 P9100..... WORKPIECE CHANGE-OVER WITH PARTING OFF MACRO)

G0 B0

T222 (MACHINING OF PART SPINDLE 2 SIDE)

G55

G92 S2500

G96 S180 M4

G0 X42 Z0 M8

...

G0 X200 Z-100

.....(UNLOAD PIECE FROM BACK SPINDLE)

M62 (INCREASE PIECE COUNTER)

M1 (OPTIONAL STOP)

IF[#1014EQ0]G0TO100 (IF THE BAR IS NOT FINISHED SKIP TO BLOCK N100)

#3000=1 (FINISHED BAR)

M30

8.4 EXAMPLE OF MACHINE TOOL BAR-FEEDER CONVEYOR USE WITHOUT BACK SPINDLE

The example below shows the use of a single pipe push-bar conveyor for machine without back spindle..

T101 (TOOL FOR FACING / PARTING OFF NEW BAR)

G54

G97 M3 S50

M10 (ENABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X0 Z5 M9

Z-47

M69 (OPEN SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

G1 Z1 F10

M68 (CLOSE SELF-CENTRING CHUCK/COLLET CHUCK)

M11 (DISABLE SPINDLE ROTATION WITH JAWS OPEN)

G0 X200 Z100

N100 T202 (COMPLETE MACHINING OF ITEM)

G54

G92 S2500

G96 S180 M4

G0 X42 Z0 M8

.....

G0 X200 Z100

T505 (PARTING OFF)

G54

G92 S2500

G96 S120 M4

G0 X42 Z-53 M8

G1 X3 F0.1

M89 (WORKPIECE UNLOADING ARM UP)

G97 S500 M4

G1 X-1 F0.05

G0 X42

M88 (WORKPIECE UNLOADING ARM DOWN)

G0 X200 Z100

M62 (WORKPIECE COUNTER INCREASE)

M1 (OPTIONAL STOP)

IF[#1014EQ0]G0TO100 (IF THE BAR IS NOT FINISHED SKIP TO BLOCK N100)

#3000=1 (FINISHED BAR)

M30

8.5 EXAMPLE OF PULL-BAR CONVEYOR USE

It is possible to carry out work from bars without using a bar loading system, using a special tool to extract the bar using the spindle axis Z .

The example below shows how this tool can be used:

N1 T202 (COMPLETE MACHINING OF ITEM)

G54

G92 S2500

G96 S180 M4

G0 X32 Z0 M8

.....

G0 X200 Z100

T505 (PARTING OFF)

G54

G92 S2500

G96 S120 M4

G0 X34 Z-32 M8

G1 X3 F0.1

M89 (PIECE UNLOADER ARM UP)

G97 S500 M4

G1 X-1 F0.05

G0 X24

M88 (PIECE UNLOADER ARM DOWN)

G0 X200 Z100 M5

M1 (OPTIONAL STOP)

T101 (PULL-BAR CONVEYOR)

G54

G94

G0 X0 Z5 M9

G1 Z-28 F2000

G1 Z-40 FF400

M69 (OPEN SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

G1 Z-8 F12000

M68 (CLOSE SELF-CENTRING CHUCK/COLLET CHUCK)

G4 U1

G1 Z5 F1000

G0 X100 Z100

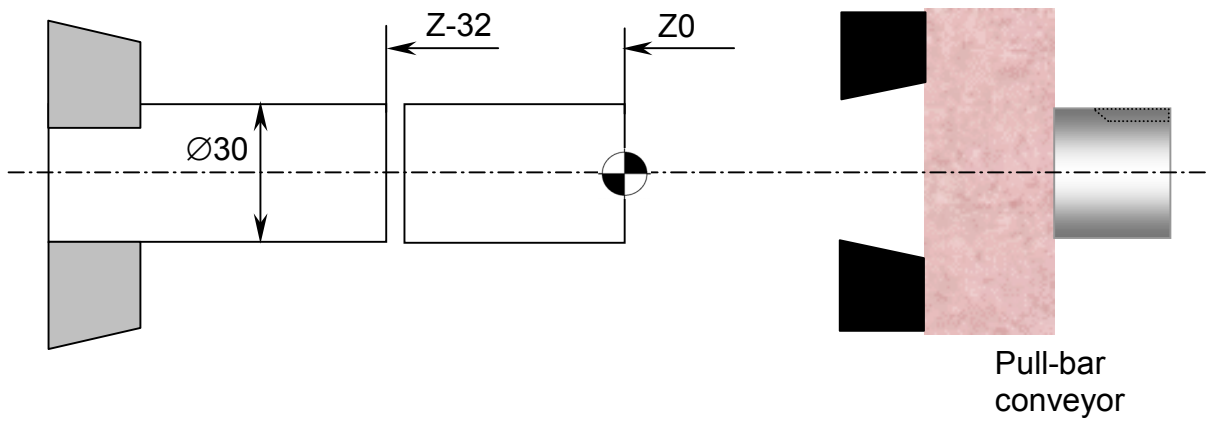
G95

M62 (INCREASE PIECE COUNTER)

GOT01

M30

All tools, including the pull-bar conveyor have to be referred to the same point (workpiece zero.)



12.0 MACHINE START-UP

Machine start-up consists in switching on and re-positioning axes, if required.

12.1 POWER-ON

To switch on the machine, follow this procedure:

- 1 - Turn the main switch, on the front panel of the machine, to 1.
- 2 - Make sure that the two emergency red keys ("mushroom") have been raised.

Wait until the power-on self test has been completed, then follow this procedure:

- 3 – Press the white **ON** key placed on the operator's panel.

The key now lights up and the machine is switched on.

For some machines, axes reference may be required before performing any other operation.

12.2 EXECUTION OF AXES REFERENCE

All axes, apart from **X**, are absolute axes. Reference points are therefore established the first time the machine is started, if a major breakdown has occurred or if the measuring system has been changed (from mm to inches)

- 1 – Press the axes reference key, placed on the operator's panel, below the screen



The led corresponding to the key of the axis that must be re-positioned will flash. This might be: **X**

Make sure that the sliding guard is closed and locked and that the axes potentiometer is active *i.e.* set on a value either than zero.

Make sure the X-axis is not positioned near the limit switch. Check for any interference that may lead to impacts, then press the keys highlighted by the leds. The machine will then re-position all the axes and, once movements are over, leds will be off whereas the machine will be ready to operate.

12.3 WRITE PROTECTION KEY

To store and to modify programmes and machine data, the protect key placed on the operator's panel is to be in horizontal position. In the remaining cases (correctors, origins, etc.) the key position is not important.

13.0 PROGRAMME MANAGEMENT

This chapter describes management of machining programmes. Management includes insertion, change and deletion of programme blocks as well as deletion, copying and renaming of programmes.

13.1 HOW TO CREATE A NEW PROGRAMME

To create a new programme, follow this procedure:

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Type in the code O followed by the desired number from 1 to 8000
- 4 - Press the key **INSERT** then press the key **EOB**
- 5 - Insert the whole programme by pressing the key **EOB** at the end of each block and the key **INSERT** to store all entered blocks.

N.B. Spaces are not required between two codes of the programme.

13.2 HOW TO MODIFY AN EXISTING PROGRAM

To modify an existing programme, follow this procedure:

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Type in the code O followed by the desired number (for ex. O8000)
- 4 - Press the soft key **RECE O**

13.3 HOW TO INSERT A CODE (OR A BLOCK) IN A PROGRAM

To insert a code (or a block) in a programme, follow this procedure:

- 1 - By means of the arrow keys, position the cursor on the previous code (if a whole block is being inserted, position the cursor on the ; "semicolon" of the previous block).
- 2 - Type in the code to be entered.
- 3 - Press the key **INSERT** (or **EOB** and **INSERT** to insert a whole block)

13.4 HOW TO MODIFY OR REPLACE A CODE

To replace or modify a code in a programme, follow this procedure:

- 1 - By means of the arrow keys, position the cursor on the code to be replaced.
- 2 - Type in the new code.
- 3 - Press the key **MODIFY**

13.5 HOW TO DELETE A CODE

To delete a code in a programme, follow this procedure:

- 1 - By means of the arrow keys, position the cursor on the code to be deleted.
- 2 - Press the key **DELETE**

13.6 HOW TO DELETE A BLOCK

To delete a block in a programme, follow this procedure:

- 1 - By means of the arrow keys, position the cursor on the block to be deleted.
- 2 - Press the key **EOB**
- 3 - Press the key **DELETE**

13.7 HOW TO COPY/PASTE PART OF A PROGRAMME

To copy/paste a number of blocks within a programme, or from one programme to another follow this procedure:

- 1 - Position the cursor on the first block to be copied
- 2 - Press the soft key **(OPER)**
- 3 - Press the soft key **+**
- 4 - Press the soft key **EDI - EX** .
- 5 - Press the soft key **COPY** .
- 6 - Press the soft key **CURS §**
- 7 - Position the cursor on the last block to be copied
- 8 - Press the soft key **§ CURS**
- 9 - Press the soft key **EXEC** .

The part of the programme that has been copied will be temporarily stored in the programme O0000.

- 10 - Position the cursor on the block following the one where the copied section is to be inserted
- 11 - Press the soft key **JOIN**.
- 12 - Press the soft key **§ CURSOR**
- 13 - Press the soft key **EXEC**

13.8 HOW TO COPY A PROGRAMME

To generate two identical programmes with different names, follow this procedure:

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Look for the programme that is to be copied (For Ex. O800 and the soft key RECE O)
- 4 - Press the soft key **(OPER)**
- 5 - Press the soft key **+**
- 6 - Press the soft key **EDI - EX** .

- 7 - Press the soft key **COPY**.
- 8 - Press the soft key **ALL**.
- 9 - Type in the new programme number (without the character O)
- 10 - Press the key **INPUT**
- 11 - Press the soft key **EXEC**.

13.9 HOW TO DELETE A PROGRAMME

To delete a programme, follow this procedure:

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Type in the address O followed by the number of the programme to be deleted
- 4 - Press the key **DELETE**

The following message will appear on the screen: DELETE O.... (number of the programme to be deleted)

- 5 - Press the soft key **EXEC** to confirm deletion of the programme

N.B. Once this procedure has been carried out, the programme following -in the list - the one that has been deleted will be automatically selected and will, therefore, be active.

13.10 HOW TO RENAME PROGRAMME

To rename a programme, follow this procedure:

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Look for the programme that is to be renamed (For Ex. O800 and soft key RECE O)
- 4 - Position the cursor on the programme number (within the programme)
- 5 - Type in the new programme number
- 6 - Press the key **MODIFY**

13.11 SELECTION OF A PROGRAMME FOR MACHINING

A programme that has been recalled to be modified or to be written is automatically active and can be used for machining purposes. The one used to modify a programme.

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Type in the code O followed by the desired number (for ex. O8000)
- 4 - Press the soft key **RECE O**

By pressing **AUTOMATIC MODE** on the operator's panel, the selected programme will be ready to perform machining.

13.12 HOW TO CREATE A NEW SUBPROGRAM

The procedure followed when creating a subroutine is similar to the one used to create a programme. Subroutines and programmes are stored in the same memory. To make management easier, values should be included between O8001 and O9000 (main programmes range from O1 to O8000). Please note that all our subroutines end up with the function M99. For further details on subroutines see programming Ready Reference.

To create a new subroutine, follow this procedure:

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Type in the code O followed by the desired number, ranging from 8001 to 9000
- 4 - Press the key **INSERT** then press the key **EOB**
- 5 - Insert the whole subroutine by pressing the key **EOB** at the end of each block; then press the key **INSERT** to store entered blocks.

N.B. Spaces are not needed between two codes of the same subroutine.

13.13 GRAPHIC SIMULATION OF A PROGRAMME

This procedure is used to display graphically (with axes and spindle at a standstill) programmed movements before running the programme in AUTO mode.

N.B. Only active programmes can be graphically displayed (for further details on how to select a programme see par. 13.11). The machine and the CNC must be switched on, the sliding guard must be open and no errors must have been detected when starting the graphic mode.

- 1 - Press the key **GRAPHIC PAGE**
 - 2 - Press **AUTOMATIC MODE** on the operator's panel
 - 4 - Press the soft key **GRAF**
 - 5 - Press the soft key **OPER**
 - 6- Press the soft key **HEAD** to rewind the program
- Be sure to have the potentiometer of axes open and not to have active machine alarms.**
- 7- Press soft key **ESEG** to start the program graphic in automatic mode or press **SINGOL PATH** to activate program graphic in singular mode.

To change graphic window dimensions proceed as below:

Press **G.PRM**

Position the cursor on **PIECE LENGHT W** insert the value in micron and press **INPUT**

Position the cursor on **PIECE DIAMETER D** insert the value in micron and press **INPUT**

To exit the graphic page, press any key on the MDI panel (**EDITING, POSITION, SETTING** etc.)

13.14 RUNNING OF THE PROGRAMME IN CYCLE

To run a selected programme in cycle, press the key **AUTOMATIC MODE** placed on the operator's panel, then press the key **RESET** if the programme has not been rewound; finally press **START** to start machining. For further details on how to run a programme in automatic mode, and on the function keys placed on the operator's panel, read paragraph 19.1

13.15 INTERRUPTION OF PROGRAMME EXECUTION

To interrupt a programme, or a function, press the key **STOP** placed on the operator's panel. You can then cancel programme execution by pressing the key **RESET**. You can either bring the programme, which is still running to the start point, or restart it from the point where it was interrupted by pressing the key **START**.

13.16 HOW TO START A PROGRAMME FROM AN INTERMEDIATE STAGE

Following this procedure, you can start the programme from an intermediate block. **WARNING!:** the following functions and addresses are not enabled, although they have been included in the previous blocks: T, G, S, M and F. This is why blocks should be looked for starting from tool calls. Origins, revolution limits and technological parameters should be redefined in the programme after this block.

To start the programme from an intermediate stage, follow this procedure:

- 1 - Select **EDIT MODE** on the operator's panel
- 2 - Press **PROGRAMME PAGE**
- 3 - Type in the code O followed by the desired number ranging from 1 to 8000
- 4 - Position the cursor on the tool call from which you want to start
- 5 - Press the key **AUTOMATIC MODE** placed on the operator's panel
- 6 - Press the green key **START** to start the programme from the selected stage.

13.17 BACKGROUND EDITING

Management of a programme when another programme is running is called background editing. The procedure is the same followed when modifying an active programme. To perform background editing, follow this procedure:

- 1 - Press **PROGRAMME PAGE**
- 2 - Press the soft key (**OPER**)
- 3 - Press the soft key **COR-BG**

A screen will appear for background editing (this will activate O0000)

- 4 - Edit the programme following the procedures described above

The programme edited in background is to be saved following this procedure:

Once the programme has been modified or written :

- 5 - Press the soft key (**OPER**)

6 - Press the soft key **FIN - BG**

14.0 TOOL RESET

Tool reset can be carried out following two different procedures described in this manual or by means of a tool-measuring probe in the machines equipped with this option.

14.1 MANUAL TOOL RESET

- 1 - Fix the rough piece on the chuck
- 2 - Press the key **MDI MODE** placed on the operator's panel
- 3 - Press the key **PROGRAM PAGE**
- 4 - Enable the first external tool to be reset followed by its offset.
For example: G54; then press **EOB INSERT START**
- 5 - Activate the first tool for external to reset followed by the corrector
example: T101 and press **EOB INSERT START**
- 6 - Put the spindle in rotation.
Example: G97 S500 M4 and press **EOB INSERT START**
- 7 - Turn the piece with **JOG Z- Z+ X- X+** controlling
with the potentiometer axes or using the hand-wheel after having selected it.
- 8 - After turning the piece move away only with Z axe
on the x co-ordinate of turning.
- 9 - Stop the spindle writing M5 and press **EOB INSERT START**
- 10- **MEASURE THE DIAMETER TURNED**
- 11- Press the soft key **PAGE SETTING till compens.**
- 12- Press the soft key **COMPEN**
- 13- Press the soft key **GEOMET**
- 14- Place the cursor on the offset to be reset
- 15- Type in X followed by the measured value (for ex. X100.3)
- 16- Press the soft key **MEASURE**
- 17- Restart spindle rotation.
For example: G97 S500 M4 then press **EOB INSERT START**
- 18- Face the piece by means of the keys **JOG Z- Z+ X- X+** controlling it with the axes potentiometer
or using a hand-wheel, after selecting it.
- 19- Once the piece has been faced, move away the X-axis only, keeping the same Z facing co-ordinate
- 20- Stop the spindle by typing in M5 then press **EOB INSERT START**
- 21- Press the key **SETTING PAGE** until the **OFFSET** window is reached
- 22- Press the soft key **COMPEN**
- 23- Press the soft key **GEOMET**

24- Position with the cursor on the corrector to reset

25- **Write Z followed by the required value**

26- press soft key MEASURE

To reset the remaining tools for external machining, repeat the procedure described above touching the previously turned diameter or stop.

14.2 CENTRE RESET.

The reset procedure in Z is similar to that used for turning tools. As to the X-axis, reset is not performed.

Proceed as follows to write Zero in the geometrical offset value of the desired corrector:

1 - Press the key **SETTING PAGE** until you reach the OFFSET window

2 - Press the soft key **COMPEN**

3 - Press the soft key **GEOMET**

4 – Go with the cursor on X of the corrector to reset

5 – Write zero

6 – Press soft key ENTRY

14.3 INTERNAL MACHINING TOOLS RESET

Once a hole has been made with the centre (unless it already exists) the procedure is similar to that followed for the first tool and the other external tools.

14.4 TOOL RESET ON COUNTERSPINDLE

Once the piece has been mounted on the counterspindle, the tool reset procedure is similar to that used for the main spindle. Make sure the procedure is started only after bringing the counterspindle axis in the position (usually zero) where machining will be performed, in the programme, entering for example "**G0 B0**" (if the machining dimension is zero) in MDI and the origin used in the programme active. In this case, machining allowance in Z is negative. For example: "**Z-0.5**" to obtain 1/2 mm. facing allowance.

14.5 RESET OF TOOLS WITH PROBE (OPTIONAL)

Reset with probe is carried out by the CNC using variables from #515 to #522. Make sure not to use these variables when programming.

To reset tools with a probe follow this procedure:

1 - Press the key **MDI MODE** placed on the operator's panel

2 - Press **PROGRAMME PAGE**

3 – Activate the first tool to be reset.

For example: T101 then press **EOB INSERT START**

4 - Press the key **PROBE EXIT** placed on the operator's panel, or programme the M238 function if a counterspindle has been chosen.

(If a Manual Probe has been chosen extract the probe arm manually)

When the probe is enabled, the CNC displays the correctors' table automatically

5 – By means of the keys JOG Z- Z+ X- X+ and checking with the axes potentiometer, position the tool near the measuring probe

6 – Reduce the potentiometer to 1 or 2 %

7 – Lean on the desired probe X+ X- Z+ Z-

Once contact has been achieved, the axis will stop automatically

8 – Move away from the probe then repeat this operation on the other axis to be reset

Repeat from item 2 to item 10 to measure another tool

All the tools have been correctly reset on the X-axis, whereas for the Z-axis, they refer to the machine "ZERO ". To refer dimensions of the Z-axis to the "piece zero point" the origin measuring procedure has to be performed (par. 4.1) with one of the tools reset on the probe.

14.6 RESET TOOLS FOR MACHINES TWIN

The proceeding to reset tools for machines Twins is equal to that of CTX. Start with this proceeding only after having brought the spindle 2 in the position in which, machining in MDI will be executed, for example "GO BO" (if quote of machining is zero)

Remember to select the work channel CN1 for spindle 1 and CN2 for spindle 2.

- 1 Mount the piece on chuck
- 2 Press MDI on operator's panel
- 3 Press PROGRAM PAGE
- 4 Select work channel with CN1/CN2
- 5 Activate the original of program editing for example G54 and press EOB INSERT START
- 6 Activate the first external tool to reset followed by the corrector ex. T101 and EOB INSERT START
- 7 Put the spindle in rotation. Ex. G97 S500 M4 and press EOB INSERT START
- 8 Turn the piece with JOG Z- Z+ X- X+ checking with potentiometer or hand-wheel
- 9 After turning the piece move only with Z axis staying on X turning co-ordinate
- 10 Stop the spindle pressing M5 and press EOB INSERT START
- 11 Measure the turned diameter
- 12 Press SETTING PAGE until COMPENS.
- 13 Press soft key COMPEN
- 14 Press soft key GEOM
- 15 Position with the cursor on the corrector to reset
- 16 Edit X followed by the measured value ex. X100.3
- 17 Press soft key MEASURE
- 18 Put the spindle in rotation. Ex. G97 S500 M4 and press EOB INSERT START
- 19 Face off the piece using JOG Z- Z+ X- X+ checking with the potentiometer

20 After the facing off of piece, move only with X axis staying on Z facing off co-ordinate

21 Stop the spindle with M5 and press EOB INSERT START

22 Press SETTING PAGE until compens.

23 Press soft key Compen

Press soft key GEOM

24 Position with the cursor on the corrector to reset

25 Edit Z followed by the desired value

26 Press MEASURE

To reset the other external tools repeat the proceeding closing as much as possible the diameter or the rabbet previously turned.

NOTE if a tool of turret 1 on spindle 2 is reset, or viceversa, the eventual overmetal on Z axis is to set as negative.

14.7 TOOL TABLE MANAGEMENT

Apart from tool reset, the tool table is also required to perform fine correction, to enter the radius of the insert and the type of tool orientation

To access the tool table, follow this procedure:

1 - Press the key **SETTING PAGE** until the OFFSET window appears on the screen

14.8 TOOL FINE CORRECTION

Once the tool table has been accessed, follow this procedure:

1 - Press the soft key **OFFSEET**

2 - Press the soft key **WEAR**

3 - Position the cursor on the X or Z-axis of the desired corrector

4 - Type in the offset value (0.1 0.15 etc.)

5 - Press the soft key **+ ENTR**

N.B. The maximum offset value for each storage is 1 mm; offset on the X-axis is diametrical.

14.9 ENTRY OF INSERT RADIUS

Entry of the insert radius is required if radius offset is being used (G41, G42, G40). Once the tool has been reset, follow this procedure:

1 - Press the key **SETTING PAGE** until the OFFSET window appears on the screen

2 - Press the soft key **OFFSET**

3 - Press the soft key **GEOMET**

4 - Place the cursor on R of the desired corrector

5 - Type in the radius value (0.4, 0.8, 1.2 etc.)

6 - Press the soft key **ENTER**

14.10 ENTRY OF TOOL ORIENTATION

Entry of tool orientation is required whenever radius offset is being used (G41, G42, G40). Once the tool has been reset, follow this procedure:

- 1 - Press the key **SETTING PAGE** until the OFFSET window appears on the screen
- 2 - Press the soft key **OFFSET**
- 3 - Press the soft key **GEOMET**
- 4 - Place the cursor on T (Type of Orientation) of the desired corrector
- 5 - Enter the type of orientation (3, 2, 8 etc.)
- 6 - Press the soft key **ENTER**

Values to be entered depend on the type of tool used, as shown in the following drawing:

14.11 ENTRY OF CUTTER RADIUS

Entry of cutter radius is required whenever radius offset is being used (G41, G42, G40) and when milling is performed in G112 or G107 mode. Once the tool has been reset, follow this procedure:

- 1 - Press the key **SETTING PAGE** until the OFFSET window appears on the screen
- 2 - Press the soft key **OFFSET**
- 3 - Press the soft key **GEOMET**
- 4 - Place the cursor on R of the desired corrector
- 5 - Enter the cutter radius (3, 5, 8 etc.)
- 6 - Press the soft key **ENTER**

15.0 ORIGIN MANAGEMENT

This procedure is used to establish one or multiple reference points, thus allowing operators to have references for the movements to be entered in the machining programme. Such references are defined as piece origin.

15.1 ORIGIN MEASUREMENT

This procedure is used to establish the piece origin when tools are reset on the probe or with an external measuring system.

- 1 – Fix the rough piece on the chuck
- 2 - Press the key **MDI MODE** placed on the operator's panel
- 3 – Call a reset tool to work position.

For example: T101 then press **EOB INSERT START**

- 4 – Start spindle rotation if needed.

For example: G97 S500 M4 then press **EOB INSERT START**

- 5 – Touch the piece origin lightly by means of the JOG Z- Z+ X- X+ keys controlling with the axes potentiometer or using a hand-wheel after selecting it
- 6 – Once the piece has been touched use JOG Z- Z+ X- X+ .
- 7 – After closed the piece, move with X axis on the co-ordinate Z of zero piece
- 8 – Stop the spindle with M5 and press EOB INSERT START
- 9 - Press setting page
- 10 – Press soft key job
- 11 – Position with the cursor on origin desired and used in the program
- 12 - Press Z quote referred to actual position
- 13 press soft key MEASURE

Once this operation has been performed, the CNC will automatically load the distance between the machine zero and the piece zero in the desired origin.

15.2 ORIGIN MODIFICATION

This procedure is used for manual modification of the piece origin used in the (origin obtained following the

procedure described in the previous paragraph)

- 1 - Press the key **SETTING PAGE** until the OFFSET window appears on the screen
- 2 - Press the soft key **MACHINE/JOB**
- 3 – Place the cursor on the relevant axis (X, Z or C axis of the desired origin)
- 4 – Enter the displacement value (for ex. 0.5)
- 5 - Press the soft key **+ENTR** for an additional shift
- 6 - Press the soft key **ENTER** for an absolute shift

NB: by **ABSOLUTE** shift we mean the insertion of a new value, whereas **ADDITIONAL** shift refers to a value to be added to an existing one.

16.0 MACHINE PARAMETERS

Machine parameters are used to fully represent the characteristics of servo-motors, as well as the specifications and the functions of the machine tool

16.1 HOW TO MODIFY A MACHINE PARAMETER

To modify a machine parameter, follow this procedure:

- 1 - Select **MDI MODE** on the operator's panel
- 2 - Press **SETTING PAGE** until the PREPARATION (MANUAL) window appears on the screen
- 3 - Write 1 (ENABL) PARAMETER WRITING
- 4 - Press the key **INPUT**
- 5 - Press **PARAMETER PAGE**
- 6 - Press the soft key **OPER** until RIC N0 appears on the screen
- 7 - Type in the number of the parameter to be modified
- 8 - Press the soft key **RIC N0**
- 9 - Write the new value to be assigned to the machine parameter
- 10 - Press the key **INPUT**
- 11 - Press **SETTING PAGE** until the window PREPARTION (MANUAL) appears on the screen
- 12 - Write 0 (DISABLE) PARAMETER WRITING

If parameters feature 8 bits, values range from bit n. 0 to bit n. 7 (from right to left) as shown in the table below:

Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
--------------	--------------	--------------	--------------	--------------	--------------	--------------	--------------

For further details on how to modify machine parameters read appendix 7 of the “ PMC Manual” included in the machine documentation

17.0 SETTING OF GT 300 TAILSTOCK

This procedure only refers to the GT300 with tailstock option, since tailstock are moved differently in the other models.

- 1 - Press the key **JOG** on the operator's panel
- 2 - Fix the piece on the chuck
- 3 - Press the key **MACRO EXECUTER PAGE**
- 4 - Press the soft key **F1 SUBSPINDLE CAMS SETTING**
- 5 - Disable, if active, the tailstock thrust by means of the selector switch **0 / 1**
ON / OFF THRUST
- 6 - Place the tailstock against the piece by means of the selector switch **1 / 2**
- 7 - Press the soft key **STORE AHEAD**
- 8 - Place the tailstock in back position by means of the selector switch **1 / 2**
- 9 - Press the soft key **STORE BACK**

Press any of the pages (**EDITING**, **POSITION**, **SETTING** etc.) to exit the TAILSTOCK SETTING macro

17.1 INSTRUCTIONS TO BE INSERTED IN THE PROGRAMME

Tailstock can be moved entering the following functions in the programme:

M22 (TAILSTOCK FORWARD)

M23 (TAILSTOCK BACK)

The tailstock thrust can be enabled or disabled in the programme by using the following functions:

M922 (ENABLE TAILSTOCK THRUST)

M923 (DISABLE TAILSTOCK THRUST)

17.2 TAILSTOCK DOUBLE SPEED OPTION

On the machines equipped with this option, the start slowing position is approximately 20 mm from the tailstock forward position. This length can be "adjusted", modifying the dimension entered in SLOWING CAM LENGTH, which can be accessed by pressing the soft key EDIT DATA. Then proceed as follows:

- 1 - Press the key **MACRO EXECUTER PAGE**
- 2 - Press the soft key **F1 TAILSTOCK CAMS SETTING**
- 3 - Press the soft key **EDIT DATA**
- 4 - By means of the arrow keys, place the cursor on **SLOWING CAM LENGTH**
- 5 - Type in the new value, for example: 10 (max value 11)
- 6 - Press **INPUT** to enter the value
- 7 - Press the soft key **ADD DATA** to store the value

Press any page (**EDITING, POSITION, SETTING** etc.) to exit the TAILSTOCK SETTING macro

NOTE: For further details on how to adjust the thrust pressure and the feed speed of the tailstock, see USER'S AND MAINTENANCE MANUAL included in the machine documents.

17.3 TAILSTOCK REPOSITIONING

If the **E78 (look for the tailstock zero)** alarm appears on the screen, follow the repositioning procedure in relation to the tailstock reference point:

- 1 - Press the key **JOG** on the operator's panel
- 2 - Press the key **MACRO EXECUTER**
- 3 - Press the soft key **F1 TAILSTOCK CAMS SETTING**
- 4 - Disable, if active, the tailstock thrust by means of the selector switch **0 / 1**
ON / OFF THRUST
- 5 - Press the soft key **FND ZERO**

Press any page (**EDITING, POSITION, SETTING** etc.) to exit the TAILSTOCK SETTING macro

18.0 TAILSTOCK AND REST MANAGEMENT ON MACHINES CTX

Some machines CTX (400,500,700) are provided with option “tailstock with automatic clamping” and option “positionnable rest”. These options management may happen manually (through selectors set on operator’s panel or with use of foot switches) or in working cycle (with functions M).

18.1 MANUAL MOVEMENT OF TAILSTOCK AND REST

To move manually tailstock and rest you need simply to make coincide the clamping index on the carriage with the corresponding place on tailstock or rest.

To make this operation, you must be in JOG have the sliding door closed, or the consent to manual commands pressed.

In case of tailstock you must be sure that the micro of positioning of quill is in the backward position and is not active the thrust through the specific selector.

In case of rest, arms must be open, in contrary case act on foot switches.

After the control of all these verifications, press the buttons to clamp tailstock and rest on operator’s panel.

TAILSTOCK CLAMPING

REST CLAMPING

Now moving the carriage of Z axis you also move the tailstock or rest blocked to it.(rapid movements are limited to 10%)

To release the tailstock or the rest from the carriage and clamp it to the bed press again the specific buttons to clamp.

18.2 INSTRUCTIONS TO INSERT IN THE PROGRAM

Rest and tailstock movement occur with the insertion in program of a list of functions M:

TAILSTOCK

M52 tailstock unclamping from the bed and clamp to the carriage

M53 tailstock unclamping from the carriage and clamp to the bed

It’s possible enable or disable from program the thrust of tailstock using the functions:

M922 enable quill thrust

M923 disable quill thrust

REST

M32 rest unclamping from the bed and clamp to the carriage

M33 rest unclamping from the carriage and clamp to the bed

M84 opening rest arms

M85 closing rest arms

In machines with option “positionnable retractable rest” you can use also the following functions:

M86 retractable rest in working position

M87 retractable rest in home position

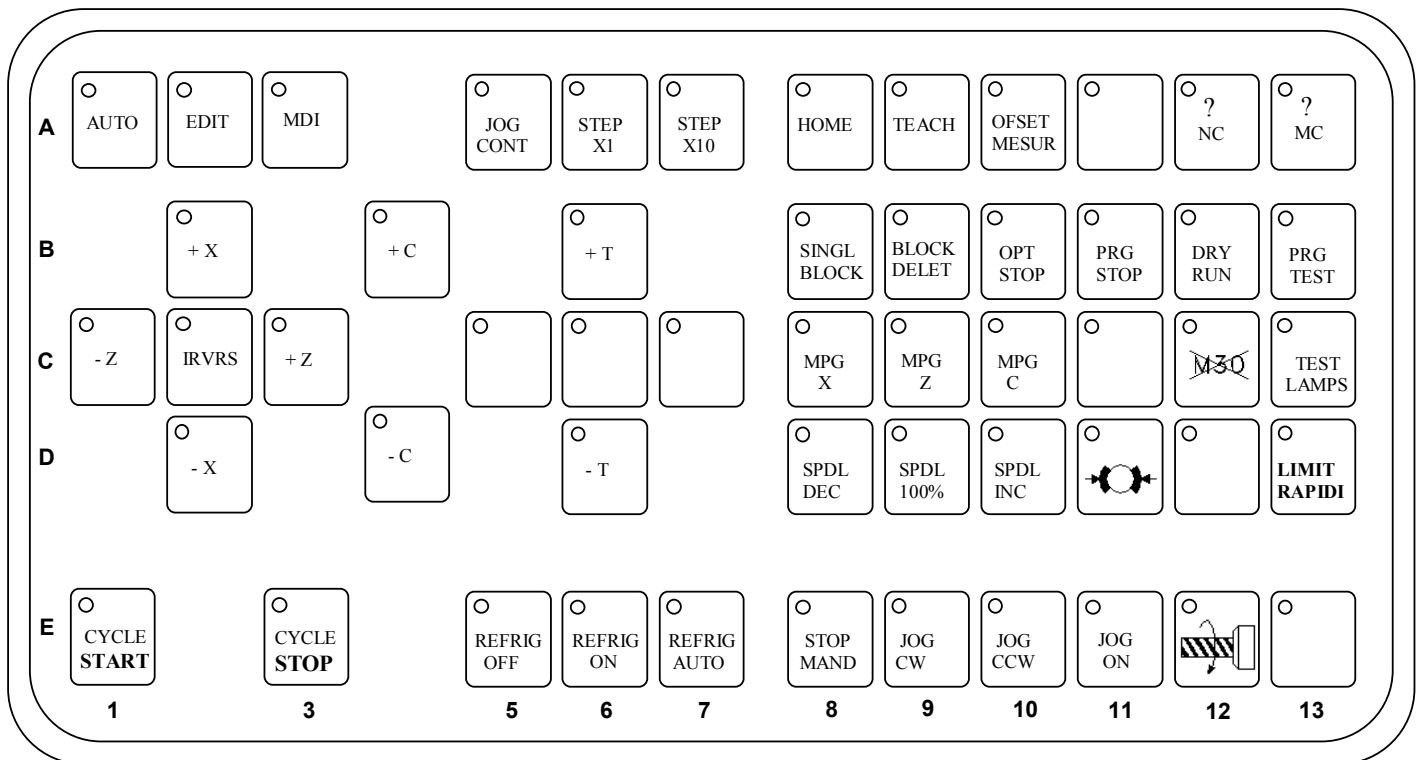
19.0 KEYBOARD AND OPERATOR'S PANEL

The CNC keys can be divided into three categories:

- Keys on the operator's panel
- Keys on the editing keyboard
- Selector switches (on the operator's panel)

19.1 KEYS ON THE OPERATOR'S PANEL

Here follows a description of the keys on the operator's panel:



AUTO (A1) AUTO MODE This key allows you to run an active programme or a graphic simulation automatically

EDIT (A2) EDIT MODE This key allows to access programme writing

MDI (A3) MDI MODE This key allows you to access the MDI page

JOG CONT (A5) This key allows you to access the JOG page

STEP X1 (A6) Incremental movement 1/1000

STEP X10 (A7) Incremental movement 1/100

N.B. By pressing **STEP X1 (A6)** and **STEP X10 (A7)** simultaneously, you can obtain incremental movement 1/10

HOME (A8) Pressing this key, axes are positioned in the reference point (see paragraph 1.2).

TEACH (A9) This Key is not used

OFFSET MEASU (A10) This key is used to select the **TOOL OFFSET** mode and to move the arm of the probe; it is only enabled if the **AUTOMATIC PROBE** option is active (in this case the probe is **MANUAL**, and it is only used to display the status):

a) By pressing this key once, the **E304** alarm will appear on the screen to verify that there are no hindrances to the movement of the probe arm.

b) By pressing this key twice, you will move the arm until the **MACHINING** position is reached and the **OFFSET** mode is automatically enabled. The **OFFSET** mode is automatically enabled in **JOG** (the offset table appears on the screen and a blinking message will indicate that any pressure on the probe will modify the active corrector) and the key led lights up.

c) By pressing this key one more time, the stationary position of the probe arm will be restored, and the led turns off.

N.B. If the probe is **MANUAL**, the key will blink when the probe is manually led to machining position. In this case the pressure on the probe arm will have no consequences.

?NC(A12) This key is used to disable alarms that do not require Resetting

?MC (A13) This key will blink any time there is a message. If an alarm has been issued, the key will not blink and an **ALM** message will appear on the status line at the bottom of the screen.

+X (B2) This key allows you to move the X-axis in + dir

- X (D2) This key allows you to move the X-axis in - dir

+Z (C3) This key allows you to move the Z-axis in + dir

- Z (C1) This key allows you to move the Z-axis in - dir

IRVRS(C2) By pressing this key in conjunction with **+X -X +Z -Z** you can speed up the movement of the selected axis

+ C (B4) This key allows you to move the C-axis in + dir

- C (D4) This key allows you to move the C-axis in - dir

+ T (B6) This key allows you to move the turret disk in + dir

- T (D6) This key allows you to move the turret disk in - dir

SINGL BLOCK (B8) By pressing this key, you can either enable or disable programme execution in a single block. If this control is enabled, press the green key **START** to process each programme block.

BLOCK DELET (B9) By pressing this key, you can enable or disable execution of blocks preceded by a slash (for example: / G0 X100 Z100 M5). If this control is enabled, the machine will not process blocks with slashes.

OPT STOP (B10) By pressing this key, you can enable or disable execution of the optional stop during machining. If this control is enabled, the machine will stop the machining process in the blocks of the programme where the **M1** function has been entered. By pressing the key **START** the machine will start the machining process from the following block.

PRG STOP (B11) By pressing this key, you can enable or disable machining of the piece, except for the M S T functions.

DRY RUN (B12) By pressing this key all machining processes are carried out at rapid speed.

PRG TEST (B13) This Key is not used

MPG X (C8) It enables the hand-wheel to move the X-axis manually

MPG Z (C9) It enables the hand-wheel to move the Z-axis manually

MPG C (C10) It enables the hand-wheel to move the C-axis manually

(C12) This allows you to use the M30 function in two different ways:



1) if this key is enabled **M30** is equivalent to **M99** (the programme will rewind and restart)

2) if this key is disabled, **M30** has the usual effect (**STOP** + programme rewinding + guard release)

The aim is to have an alternative to the use of blocks preceded by slashes, enabling a complete programme to perform both continuous machining and machining of a single piece, pressing one key and without modifying the programme itself.

TEST LAMPS (C13) This key is used to check efficiency of the leds on the operator's panel

SPDL DEC (D8) This key is used to reduce, by increments of 10%, the number of programmed spindle revolutions, until a minimum of 50% is reached

SPDL 100 % (D9) This key is used to restore the number of revolutions programmed for the current spindle (100%)

SPDL INC (D10) This key is used to increase, by increments of 10%, the number of programmed spindle revolutions, until a maximum of 120% is reached



(D11) SPINDLE BRAKE By pressing this key, you can enable or disable the brake on the spindle. This key is only enabled on machines equipped with the **"C-AXIS"** option.

RAPID LIMIT (D13) By pressing this key you can enable or disable the limit of the axes rapid movement, setting a value equivalent to 20% of the maximum available value (axes potentiometer is only enabled below 20%).

Work feed is equivalent to the programmed feed. You can modify it by means of the axes potentiometer. If this key is enabled, the led will start blinking.

CYCLE START (E1) By pressing this key, the active programme is started or the selected MDI block is processed.

CYCLE STOP (E3) By pressing this key, the cycle is stopped together with the axes. By pressing cycle start (E1), the cycle and the axes will start again.

COOL OFF (E5) By pressing this key, coolant is no longer supplied to tools

COOL ON (E6) By keeping the key pressed, coolant will be supplied to tools. This key is used during machine tooling, to make sure that outlet nozzles are correctly oriented

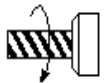
COOL AUTO (E7) By pressing this key you can enable or disable coolant supply to tools. Of course, M7 and M8 must be entered in the current programme.

STOP SPIND (E8) **By pressing this key, you can stop the spindle together with the cycle: only the spindle can be restarted by pressing the key E8 one more time (the axes and the cycle will not start); by pressing cycle start, the cycle can be restarted.**

JOG CW (E9) **This key enables manual clockwise rotation of the spindle.**

JOG CCW (E10) **This key enables manual anti clockwise rotation of the spindle.**

JOG ON (E11) **If this key is enabled, the spindle will rotate even if the keys jog cw and jog ccw keys have been released; on the contrary, if JOG ON is disabled, the spindle will stop if the jog cw or jog ccw keys are released.**



(E12) JOG UT MOT **This key is used to select the jog keys of the spindle to be used with the rotating tool. This key is only enabled on the machines equipped with the “C-AXIS” option.**

The operator’s panel also includes:

AXES POTENTIOMETER **This selector switch allows you to vary the feed speed and the rapid speed of the axes from a minimum of 0% to a maximum of 120%**

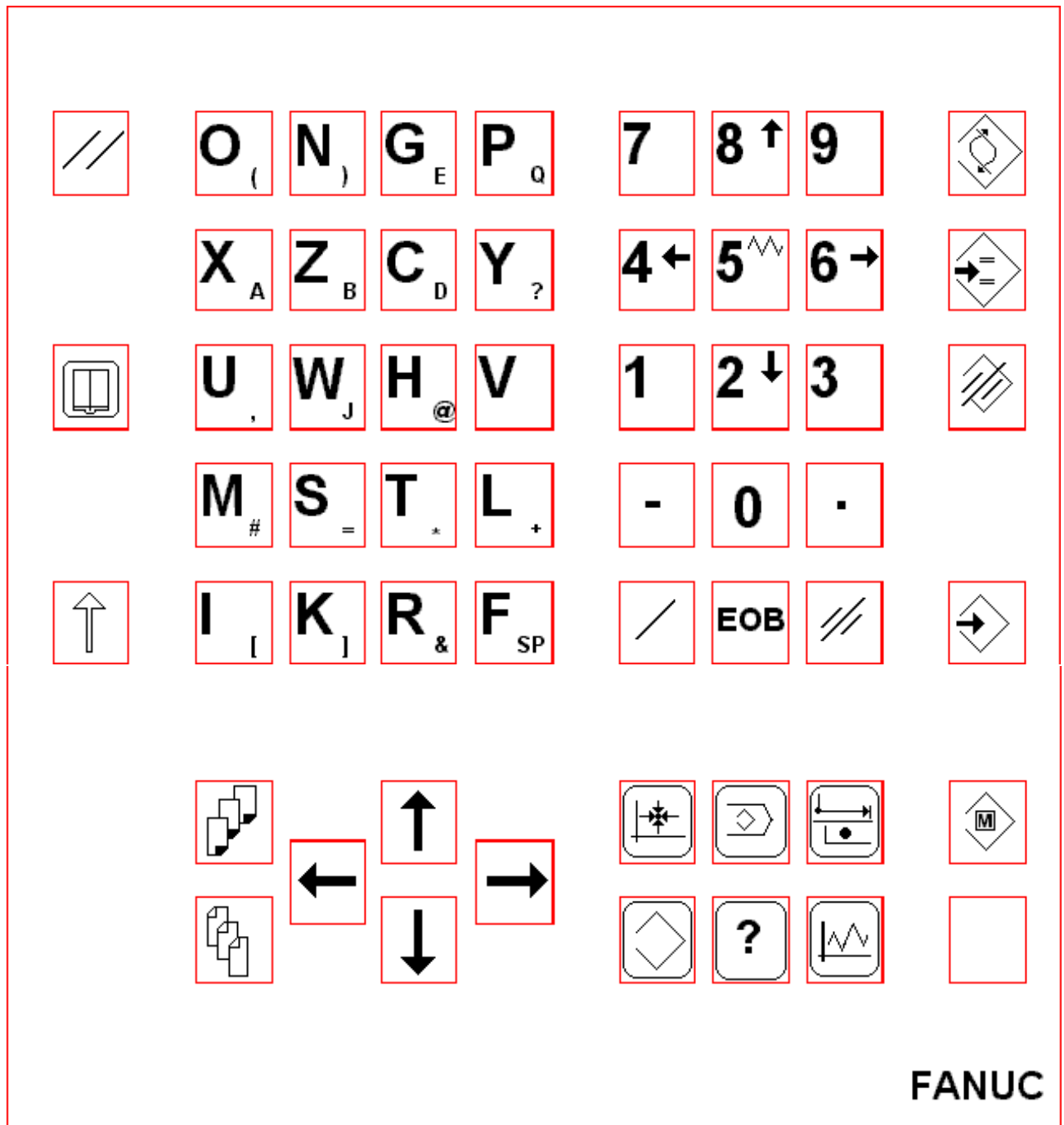
HAND-WHEEL **Once it has been enabled using the specific keys, it allows you to move the X, Z and C axes manually, with a 0,001 mm., 0,01 mm. or 0,1 mm pitch.**

EMERGENCY MUSHROOM SOFT KEY **By pressing this mushroom soft key you can switch off the machine, whereas the CNC stays on.**

PROGRAMME WRITE PROTECTION KEY **To store or modify programmes and machine data, the protection key placed on the operator’s panel must to be in horizontal position. In the remaining cases (correctors, origins, etc.) the key position is not relevant.**

19.2 KEYS ON THE MDI PANEL

Here follows a description of the keys on the MDI operator's panel :





RESET KEY Press this key to reset the CNC or to cancel alarms



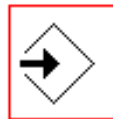
HELP KEY Press this key whenever you have doubts on one of the keys on the MDI panel or on the meaning of a CNC alarm.

SOFT KEYS Soft keys may have different functions, depending on the applications. Their function is displayed at the bottom of the screen

ADDRESS KEYS AND NUMBER KEYS Press these keys to enter alphanumeric characters or special characters.



SHIFT KEY Some address keys correspond to two characters. The key SHIFT allows you to switch between these two characters. When the character at the bottom right is enabled, the symbol ^ is displayed in the relevant line.



INPUT KEY The data entered by means of the keyboard are stored in the keyboard buffer memory and displayed. To move the content of the keyboard buffer memory to the desired line, press the key INPUT. This key is equivalent to the soft key ENTER. By pressing one of these keys, you will have the same result.



DELETEKEY Press this key to delete the last character stored in the keyboard buffer memory.

EDITING KEYS

The programme features three editing keys:



MODIFY (to modify the value of one code or to replace it with another one)



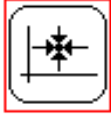
INSERT (to insert a new code)



DELETE (to delete a code)

FUNCTION KEYS

There are seven function keys referred to as pages:



POSITION PAGE: This page is used to display absolute, relative and machining dimensions of the current machining process and of the manual shift.



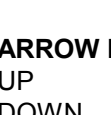
PROGRAMME PAGE : This page is used to manage programmes in EDIT MODE (*i.e.* you can write, modify or delete programmes) and to write the codes for the MDI MODE execution.



SETTING / OFFSET PAGE: This page is used to display and modify tool offset, origins and machine parameters that can be accessed by operators.



PARAMETERS PAGE : This page is used to display and modify all machine parameters.



MESSAGES AND ALARMS PAGE : This page is used to display the codes and the texts of all the alarms



GRAPHIC PAGE: This page is used to perform graphic simulation of the active programme (Par 2.12)



MACRO EXECUTER PAGE: Manufacturers can use this page to create customised macros for specific options (for ex: movement of the GT300 tailstock)

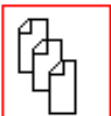
ARROW KEYS The movement of the cursor is controlled by four keys:

UP
DOWN
RIGHT
LEFT

PAGE KEYS To skip from one page to another, the following keys can be used :



replaces the current page with the following one



PAGE DOWN replaces the current page with the preceding one.

19.3 SELECTOR SWITCHES AND KEYS BELOW THE OPERATOR'S PANEL

These keys are located below the operator's panel; you can use them to enable or disable (depending on the options) different machine functions.



MANUAL CONTROLS ENABLED By keeping this key pressed, machining can be carried out in JOG MODE or in MDI MODE even if the sliding guard is open (500 spindle revolutions max. and rapid speed at 20%).



GUARD RELEASE By pressing this key you can release the sliding guard. (this control is automatically enabled in cycle by the M30, M0 and M1 functions)



CYCLE START By pressing this key the active programme can be started or the selected MDI block can be processed.



AUTOMATIC SLIDING GUARD On the machines equipped with this option, the selector switch can be used to open / close the automatic sliding guard.

TAILSTOCK MOVEMENT On the GT300 machines equipped with this option, the selector switch can be used to move the tailstock back/forward

TAILSTOCK THRUST By pressing this key, the tailstock thrust can be enabled or disabled. This control is only enabled on the machines equipped with a tailstock



TAILSTOCK HOOK UP On the machines equipped with this option, this key is used to hook up / release the tailstock to the slide. Hook up can only be performed if the tailstock reference notch and the slide reference notch coincide.



STEADY REST HOOK UP On the machines equipped with this option, you can press this key to hook up/release the steady rest of the trolley. Hook up can only be performed if the steady rest reference notch and the trolley reference notch coincide



MANUAL CONTROLS ENABLING KEY This key is used to select the MACHINING / TOOLING mode.



LAMP This selector switch is used to choose whether the tooling zone of the machine is to be lit up or not



MACHINE POWER ON This key is used to turn on the machine (see Par. 12.1)

20.0 COMMUNICATION ON SERIAL PORT

FANUC is equipped with a serial port complying with the RS232C standards. It can be used to communicate both with “intelligent” peripherals (e.g. computers) and with “dummy” devices (such as printers, tape recorders etc.).

The parameters of the serial port and the connection diagram is described below.

20.1 SETTING OF DATA TRANSFER PARAMETERS

To transfer data by means of the RS232C serial port configure transmission parameters as follows:

(to modify a parameter machine see chapter 5)

PARAMETER 020 = 0 (I/O CHANNEL) selects the type of peripheral

Once changed the parameter it's necessary go to a specific table to set the data transfer parameters.

access to this table proceed as below:

1 select MDI MODE on operator's panel

2 press PARAMETER PAGE

3 press soft key +

4 press soft key +

5 press soft key +

6 press soft key TUN IO

The table is the following.

CHANNEL I/O	TV CHECK = OFF
DEVICE NUM. = 0	PUNCH CODE= ISO
BAUDRATE = 9600	INPUT CODE = ASCII
BIT STOP = 2	FEED OUTPUT = LF
NULL INPUT (EIA) NO	EOB OUTPUT = LF
TV CHECK (NOTES) = OFF	

20.2 CABLE SCHEME RS232C

After having established the data transfer parameters, it's necessary to built a cable:

STANDARD CONNECTION

Connector 9 pin side PC	connector 25 pin side CN
Connect pin 1-4-6 and the pin 7 with 8	connect pin 6-8-20 and pin 4 with 5
RXD 2	2 TxD
TxD 3	3 RxD
GND 5.....	7 GND

COMPLETE CABLE CONNECTION (7wires)

CONNECTOR SIDE PC

9 places female

CONNECTOR SIDE CNC

25 places male

RxD 22 TxD

TxD 3.....3 RxD

DTR 46 DSR

GND 57 GND

DSR 6.....20 DTR

RTS 7.....5 CTS

CTS 8.....4 RTS

8 CD

POSSIBLE CONFIGURATIONS OF CONNECTORS CONNECTION

Connector PC

Connector CN

9 PIN	25 PIN	25 PIN	20PIN CONNECTOR JD36A (internal)
pin 5	pin 7.....	pin 7	pin 16 (mass –mass)
pin 3	pin 2.....	pin 3	pin 1 (transmission-reception)
pin 7	pin 4.....	pin 5	pin 5 (RTS-CTS)
pin 6	pin 6.....	pin 20	pin 13 (DSR-DTR)
pin 2	pin 3.....	pin 2	pin 11 (reception- transm.)
pin 8	pin 5.....	pin 4	pin 15 (CTS-RTS)
pin 4	pin 20.....	pin 6	pin 3 (DTR-DSR)
		pin 8	pin 7 (CD)

20.3 TRANSFER PROGRAMS

Here are some transfer programs tested on Graziano SPA machines with CN Fanuc.

Suggested devices have been tested with a Graziano machine with PC Windows 95 have been tested many types of software of communication, with cable long 10 meters.

Bigger distances are often possible but they are connected to the cable quality, connectors quality, and serial port of PC used.

CONNECTION WITH HYPER WINDOWS TERMINAL

This software is an option included in windows operative system, insert the following configurations:

SERIAL PORT PC

Propriety: COM 1

Bit at second : 9600

Data bit: 8

Parity: none

Stop bit: 2

Flux control : none

PROGRAM SETTINGS

Terminal buttons

Emulation: **Auto detect**

Number of buffer lines of backward sliding: 10

SETTING ASCII

TRANSMISSION

Add feed (LF) at every return to start (CR) sent.

RECEPTION

Add feed (LF) at every return to start (CR) sent.

Return automatically.

Proceeding:

To receive a file select the voice "TRANSFER" and "CAPTURE TEXT" (CN-PC), edit the path and the name with which you want to save the program, and START.

To transfer a file select the voice "TRANSFER" and "SEND FILE OF TEXT" (PC-CN), write the path and the name of program to transfer and select OPEN. The beginning of transmission is underlined by character `^`, the end is put in evidence by character `!!`.

NOTE. If during the load of a program in the CN, the key doesn't find in the right position (open memory), is visualised the alarm "071 Not find data" and the program is not loaded in memory.

To end the communication select "FILE", "EXIT", now a window appears with question "connection now, you want to exit?" answer "Yes".

For a modification or reading of the program open with editor Winword; at the end, save always the mode "only text".

CONNECTION WITH CDS SOFTWARE

This software is the program of communication standard for Graziano controls produced by PHILIPS /HEIDENHAIN (432,532,Pilot 1150)

NAME = FANUC

PORT = 1

PROTOCOL = PUN

SPEED = 9600

CODE CHARACTER = ASCII

TIME DELAY = 10

CNC VERSION = V200

NOTIFY = N

NOTE: Press CTRL +PAUSE to interrupt the program at the end of reception and transmission.

CONNECTION WITH SOFTWARE RS232

This program only works in mode DOS of PC.

DEVICES

PC COM 1:9600,E,7,2

CONNECTION WITH SOFTWARE V24

DEVICES

PROTOCOL = FANUC

PORT = COM 1

BAUDRATE = 9600

NULLFILTER = yes

BITS DATA = 8

END COMMUNICATION = TIMEOUT

BIT STOP = 2

EXTENSION = DAT

PARITY = AUS

HANDSHAKE = AUS

20.4 HOW TO COPY A PROGRAMME ON MEMORY CARD

To transfer a programme from the CNC memory to the RS 232C serial port, proceed as follows:

- 1 - Connect the memory card to that of Pc
- 2 - Press the key **EDITING MODE** on the operator's panel
- 3 - Press PROGRAM PAGE
- 4 - Write the code 0 followed by desired program number (ex. 08000)
- 5 - Press the soft key +
- 6 - Press soft key WRITE
- 7 - Press the soft key **EXEC**

20..5 HOW TO COPY A PROGRAMME FROM THE SERIAL PORT

The following procedure is used to transfer a programme from the PC to the CNC memory.

- 1 - Connect the machine serial port to the memory of CN to that of PC
- 2 - Press the key **EDITING MODE** on the operator's panel
- 3 - Press PROGRAM PAGE
- 4 - Write O code followed by desired program number (ex. 08000)
- 5 - Press the soft key +
- 6 - Press soft key READ
- 7 - Press the soft key **EXEC**

20.6 HOW TO COPY A PROGRAMME ON THE MEMORY CARD

If you want to use the Memory Card to transfer and receive programmes, you first need to configure machine parameter n. 20 a 4 (to modify a parameter see chap. 5)

- 1 - Insert MEMORY CARD in the opening on left side of monitor
- 2 - Press the key **EDITING MODE** on the operator's panel
- 3 - Press PAGE PROGRAM
- 4 - Write code O followed by desired program number (ex. 08000)
- 5 - Press the soft key +
- 6 - Press the soft key WRITE
- 7- Press soft key EXEC

20.7 HOW TO COPY A PROGRAMME FROM THE MEMORY CARD

If you want to use the Memory Card to transfer and receive programmes, you first need to configure machine parameter n. 20 a 4 (to modify a parameter see chap. 5)

- 1 - Insert the MEMORY CARD in the opening on left side of monitor
- 2 - Press the key **EDITING MODE** on the operator's panel
- 3 - Press key PROGRAM PAGE
- 4 - Write code O followed by desired program number (ex. 08000)
- 5 - Press the soft key +
- 4 – Press soft key READ
- 7 – Press soft key EXEC